



BricsCAD V14

User Guide

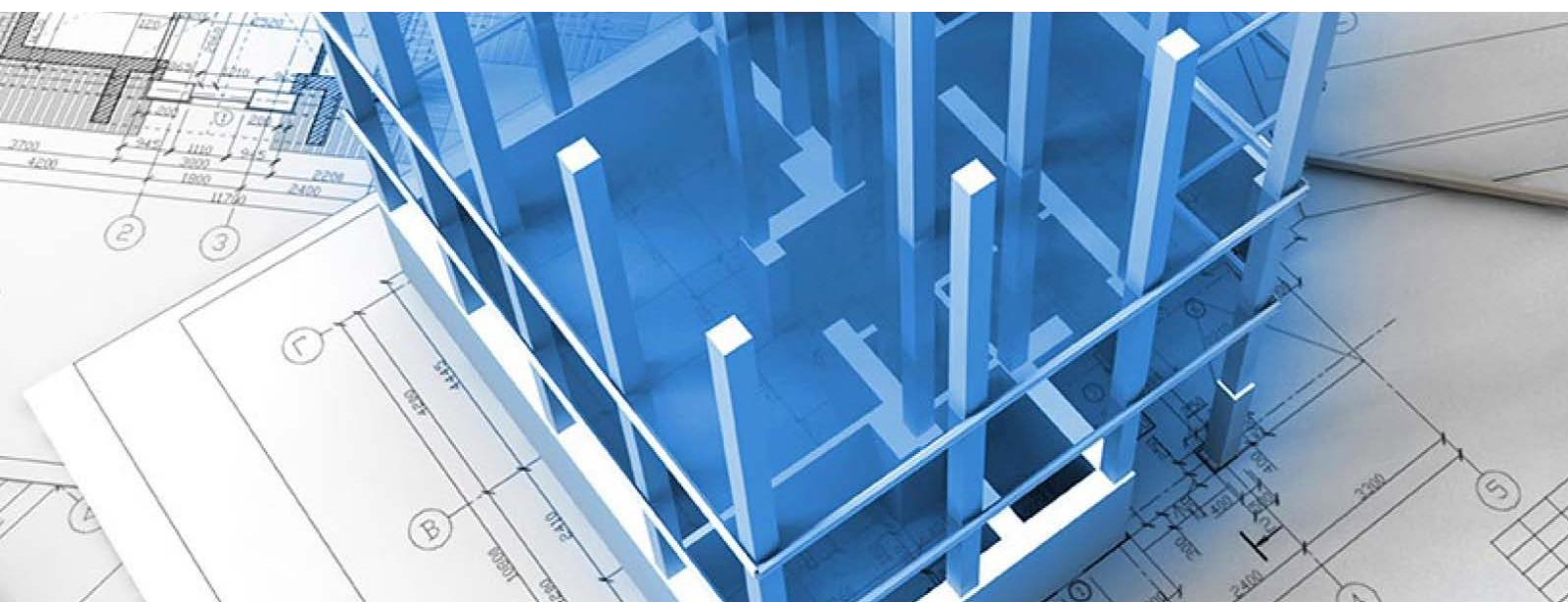


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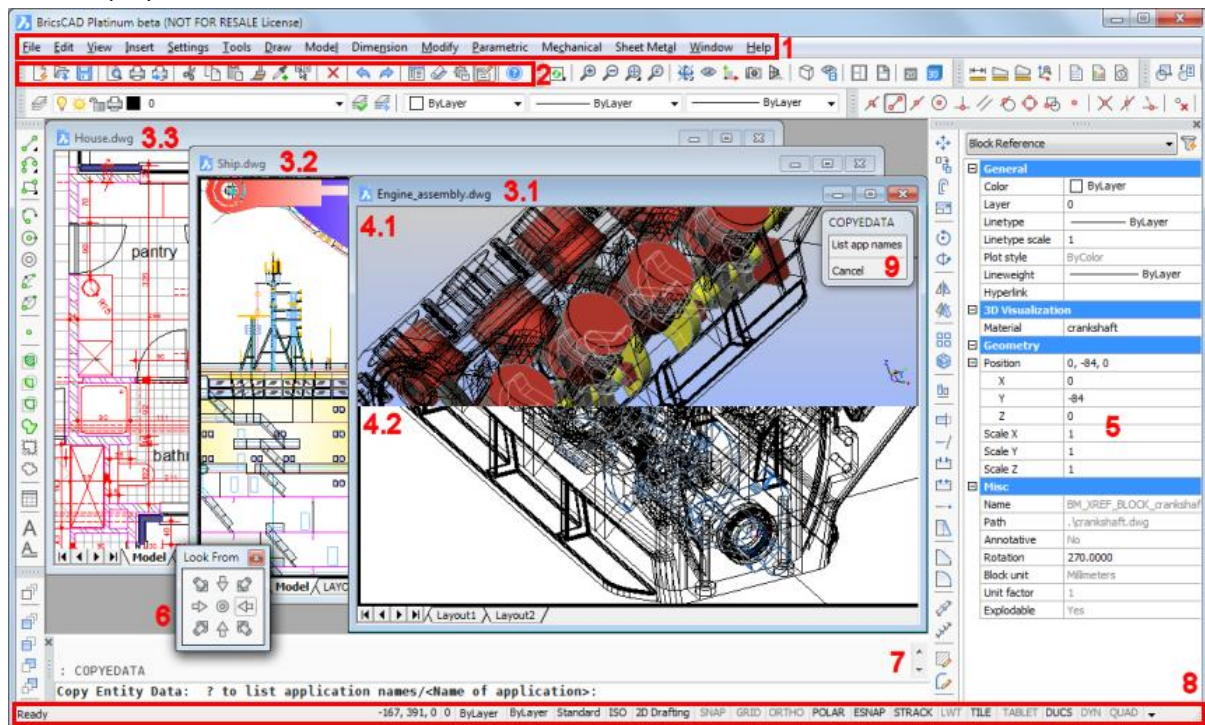
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The BricsCAD Application Window

The layout of the BricsCAD application window can be fully customized.

You can:

- open / close the command window
- open / close the status bar
- customize the status bar
- modify drawing settings in the status bar
- display / hide scroll bars



Components of the BricsCAD application window:

1. Menu Bar
2. Toolbar (docked)
3. Drawing windows
4. Drawing viewports
5. Properties Bar
6. Toolbar (floating)
7. Command bar
8. Status Bar
9. Prompt Menu

Open / Close the command bar

Do one of the following:

- Choose *Command Bar* in the *View Menu*.
- Move the cursor to a docked toolbar, then right click and choose *Command Bar* in the context menu.
- Double click the *Status* field at the left hand side of the *Status Bar*.

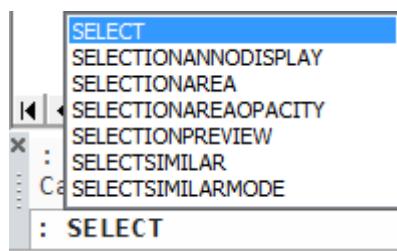
The *Command Window* closes if it was open and vice versa.



1. Close button
2. Press and hold the left mouse button to move the command bar
3. *Command History List* field: the content of the history list field can be copied to the command edit field.
4. *Command Edit* field
If the Enable flag of the AUTOCOMPLETEMODE system variable is set, command entries are completed automatically.

Do one of the following:

- Press Enter if the currently completed command displays.
- Choose a command in the *Suggestion list*.



Autocomplete options are:

<input type="checkbox"/> Auto complete mode	0x000F (15)
0x0001	<input checked="" type="checkbox"/> Enable
0x0002	<input checked="" type="checkbox"/> Auto-Append
0x0004	<input checked="" type="checkbox"/> Suggestion List
0x0008	<input checked="" type="checkbox"/> Display Icons (unsupported)
0x0010	<input type="checkbox"/> Exclude the display of System Variables
Auto complete delay	0.3

5. Command history scroll buttons





NOTES

- You can modify the height of the docked command window by dragging its top edge, when docked *Lower* or bottom edge, when docked *Upper*.
- The size of a floating command bar can be adjusted by dragging one of its edges.
- When the command bar is closed, command options and keyboard entries display in the status bar.
- Use the arrow-up and arrow-down keys of the keyboard to browse the command history.

To set the command line properties

1. Open the Settings window.

2. Go to *Program Options / User Preferences / Command Line*.

<input checked="" type="checkbox"/> Command line	
Command line font name	Verdana
Command line font size	8
Command line list background color	 (255,255,221)
Command line list foreground color	 (0,0,0)
Command line edit background color	 (248,248,248)
Command line edit foreground color	 (0,0,0)
Command line state	Command line is visible
Prompt prefix	:
Scroll history	256
Text evaluation	All responses to prompts for text strings and attribute values are taken literally

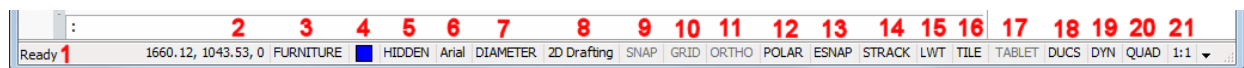
Open / Close the status bar

Do one of the following:

- Choose *Status Bar* in the *View* Menu.
- Press the *Shift + F3* keyboard shortcut.

The *Status Bar* closes if it was open and vice versa.

Working with the status bar

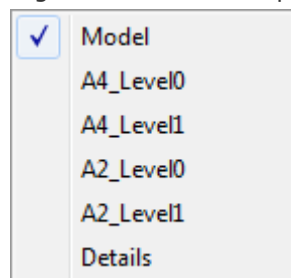


The *Status Bar* fields are:

1. Status:
 - displays the status of the software.
 - when the cursor is in a menu or on a toolbar: gives a brief description of the tool or menu item.
 - when the command window is closed: displays the tool options and keyboard entry.
 - double click to open/close the *Command Bar*.
2. Coordinates: displays the coordinates of the current cursor position (depending on the setting of the COORDS variable).
 - Right click, then choose *Off*, *Relative* or *Absolute* in the context menu.
 - Click to cycle the *Coordinate* setting. The sequence is *Off*, *Relative*, *Absolute*.
3. Layer: displays the name of the current layer.
 - click to open the *Layer Explorer*.
 - right click to select the current layer
4. Color: displays the current color.
 - click to open the *Select Color* dialog window.
 - right click to select one of the basic colors.
5. Linetype: displays the name of the current linetype.
 - click to open the *Linetype Explorer*.
 - right click to select the current linetype
6. Text Style: displays the name of the current text style.
 - click to open the *Styles Explorer*.
 - right click to select the current text style.
7. Dimension Style: displays the name of the current dimension style
 - click to open the *Dimensions Settings* dialog window.
 - right click to select the current dimension style.
8. Work Space: Indicates the current workspace.
 - click to set the value of the WSCURRENT system variable in the command bar.
 - right click, then select the current workspace

9. Snap:
 - click to toggle *Snap* on (**SNAP**) / off (**SNAP**).
 - right click, then choose *Settings* to edit the *Snap and Grid* settings.
10. Grid: click to toggle the display of *grid points* on (**GRID**) / off (**GRID**) in the current viewport.
11. Orthogonal Mode: click to toggle *Orthogonal mode* on (**ORTHO**) / off (**ORTHO**).
Turning *Orthogonal mode* on automatically disables *Polar Tracking*.
12. Polar Tracking (AutoSnap):
 - click to toggle *Polar Tracking* on (**POLAR**) / off (**POLAR**).
Turning *Polar Tracking* on automatically disables *Orthogonal mode*.
 - right click, then choose *Settings* to edit the *Polar Tracking* settings.
13. Entity Snaps:
 - click to toggle *Entity Snaps* on (**ESNAP**) / off (**ESNAP**).
 - right click, then choose *Settings* to edit the *Entity Snaps* settings.
14. Snap Tracking (AutoSnap):
 - click to toggle *Snap Tracking* on (**STRACK**) / off (**STRACK**).
 - right click, then choose *Settings* to edit the *Snap Tracking* settings.
15. Line Weight Display:
 - click to toggle the display of *Line Weights* On (**LWT**) or Off (**LWT**).
 - right click, then choose *On* or *Off* to control the display of *Line Weights*.
16. Viewport Mode:
 - In the *Model* layout:
Tile : Model space with tiled viewports.

Click to switch to the most recently opened paper space layout.
Right click to select a paper space layout:

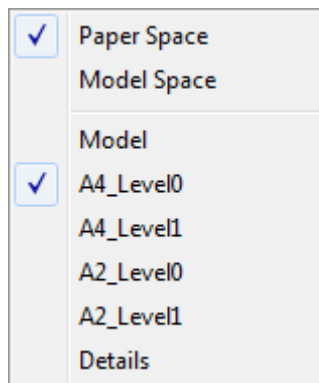


- In a *Layout*:
M:Layout : Model space with floating viewports
P:Layout : Paper space

Click to toggle between paper space and model space with floating viewports.
Right click, then choose:

- Paper Space: Switch to paper space
- Model Space: Switch to model space with floating viewports
- Model: Switch to model space with tiled viewports

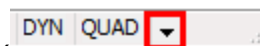
- Layout: Switch to the selected paper space layout.




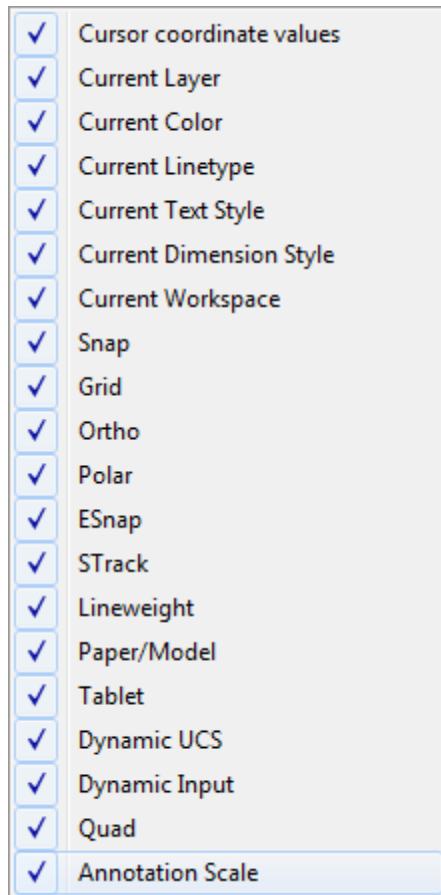
17. Tablet: Initializes the use of a drawing tablet.
18. Dynamic UCS: click to toggle the Dynamic UCS feature ON (**DUCS**) or Off (**DUCS**).
19. Dynamic Dimensions:
 - click to toggle the display of *Dynamic Dimensions On* (**DYN**) or Off (**DYN**).
 - right click, then choose *Settings* to edit the *Dynamic Dimensions* settings.
20. Quad Display:
 - click to toggle the display of the *Quad On* (**QUAD**) or Off (**QUAD**).
 - right click, then choose *Settings* to edit the *Quad* settings.
21. Annotation Scale:
 - displays the current annotation scale (CANNOSCALE system variable).
 - right click, then choose an annotation scale in the list.
Choose *Custom...* in the list to launch the SCALELISTEDIT command.

NOTE *Orthogonal mode* (field 10) is switched off if *Polar Tracking* (field 11) is on and vice versa.

To customize the status bar



1. Click the *down arrow* button () at the right hand side of the *status bar*.
The status options display.

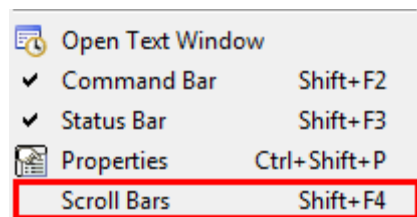


2. Do one of the following:
 - Click an unmarked field name to add it to the status bar.
 - Click a marked field name to remove it from the status bar.

Display or hide scroll bars

To toggle scroll bars on / off:

1. (option) Hold down the Shift key, then press the *F4* function key.
2. (option) Choose *Scroll Bars* in the *View* menu.







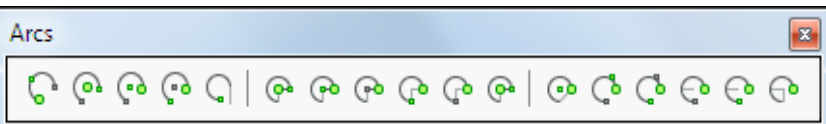
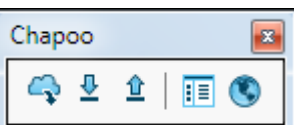
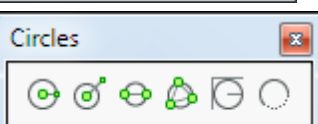
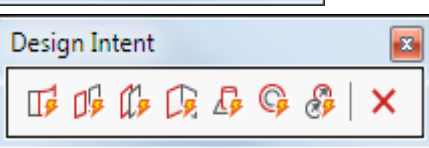

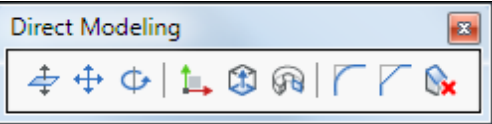

3. (option) Type *scrollbar* in the command bar, then do one of the following:
 - type *off* in the command bar or choose *Off* in the context menu.
 - type *on* in the command bar or choose *On* in the context menu.
 - type *T* in the command bar or choose *Toggle* in the context menu.

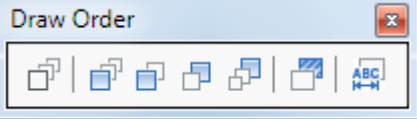
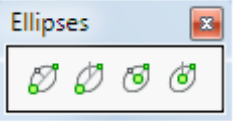
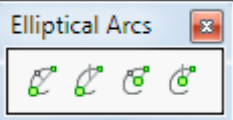
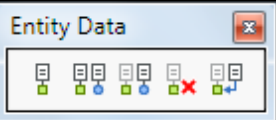





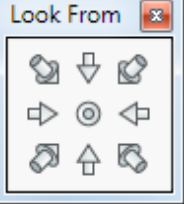
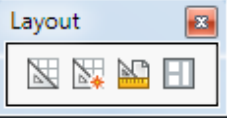

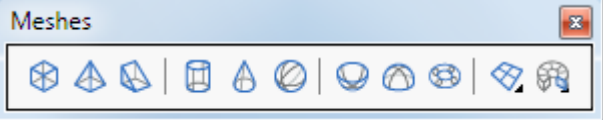
Toolbars










The following toolbars are available in BricsCAD.

Some of the tools are available in BricsCAD Pro and/or BricsCAD Platinum only. Please see the Command Reference guide for more information.

(*) These toolbars are new in BricsCAD V14.

2D Constraints	
3D Constraints	
3D Solid Editing	
3D Solids	
Arcs	
Chapoo (*)	
Circles	
Design Intent	
Dimensions	
Direct Modeling	
Draw	

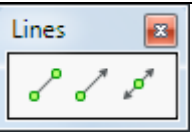

Draw Order	 The Draw Order toolbar contains icons for moving objects to the front, back, or top/bottom of the stack, and a button to toggle the Draw Order dialog.
Ellipses	 The Ellipses toolbar contains four icons for creating different types of ellipses: a full ellipse, a partial ellipse, an arc, and a segment.
Elliptical Arcs	 The Elliptical Arcs toolbar contains four icons for creating different types of elliptical arcs: a full ellipse, a partial ellipse, an arc, and a segment.
Entity Data	 The Entity Data toolbar contains icons for creating and editing entity data, including a button to open the Entity Data dialog.
Entity Properties	 The Entity Properties toolbar contains icons for setting entity properties such as color, layer, and visibility. It includes a dropdown menu for color and a checkbox for 'ByLayer'.
Entity Snaps	 The Entity Snaps toolbar contains icons for enabling and disabling various snapping modes, including point, endpoint, midpoint, and center.
Images	 The Images toolbar contains icons for inserting and editing images, including a button to open the Image Properties dialog.
Insert	 The Insert toolbar contains icons for inserting various objects, including text, images, and blocks.
Inquiry	 The Inquiry toolbar contains icons for performing various types of inquiries, including distance, area, and volume.
Look From	 The Look From toolbar contains icons for setting the view direction, including buttons for top, bottom, left, right, and isometric views.
Layout	 The Layout toolbar contains icons for managing the layout, including buttons for creating and deleting layouts.
Mechanical	 The Mechanical toolbar contains icons for creating and editing mechanical parts, including buttons for creating and deleting parts.
Meshe s	 The Meshe s toolbar contains icons for creating and editing meshes, including buttons for creating and deleting meshes.

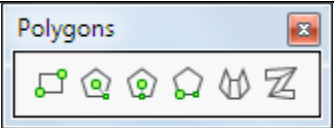
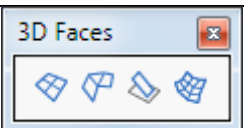
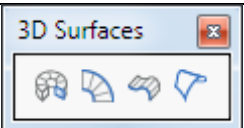
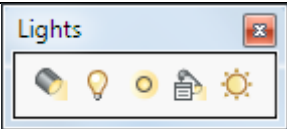


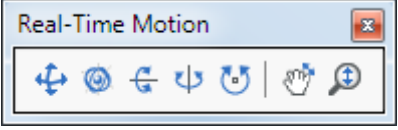
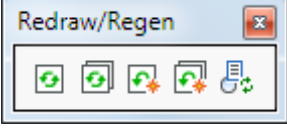
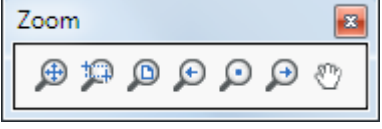
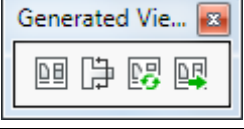
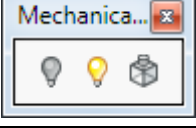
Modify	
Ref Edit	
Rendering	
Sections	
Settings	
Sheet Metal (*)	
Standard	
Tools	
View	

Toolbar Flyouts

On some tool buttons a small black arrow at the bottom right corner of the icon indicates a flyout is available. A flyout holds a group of related tools. Press and hold the left mouse button to expand the flyout and choose one of the flyout tools. The tool that was last chosen remains visible in the collapsed toolbar.

Each of the flyouts can be opened as a separate toolbar.

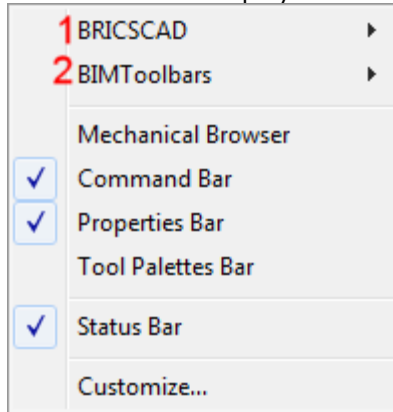
Parent	Flyout	
Draw	Lines	
	Polylines	

	Polygon		
Meshes	3D Faces		
	3D Surfaces		
Rendering	Lights		
Settings Entity Properties	Layers		
Tools	Attributes		
View	Real-Time Motion		
	Redraw/Regen		
	Zoom		
	Generated Views		
Mechanical	Mechanical Visualization		

Opening a toolbar

1. In *Windows*: Hover over a toolbar or a blank portion of the toolbar area, then right click.
In *Linux*: Hover over a blank portion of the menu bar, status bar or toolbar area, then right click.

A context menu displays.



(1) Main CUI file.

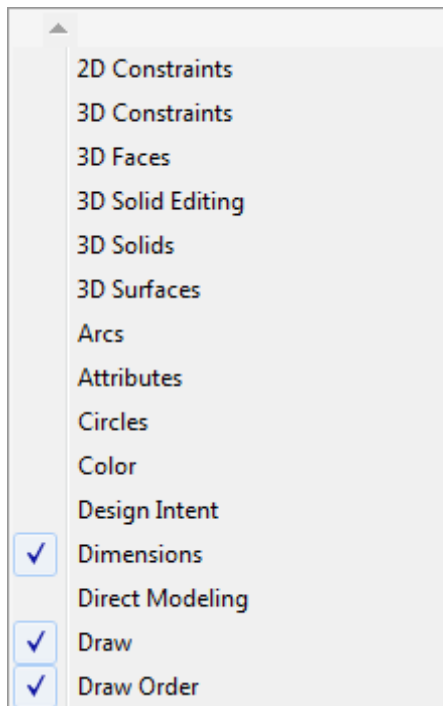
(2) Partial CUI file(s)

2. In the context menu choose either the Main CUI file (e.g. BRICSCAD) or one of the Partial CUI files (if any).


A list of available toolbars in the selected CUI file displays.

Toolbars that are already open are checked.

If the list does not fit on the screen, click the arrow keys on the top and the bottom to scroll.




3. Select the toolbar on the context menu.
The toolbar opens.
4. (option) Dock the toolbar by dragging the toolbar by its title bar to one of the edges of the BricsCAD application window.

- NOTES**
- If you select a toolbar that was already open, it will be closed.
 - To prevent a toolbar from docking, press and hold the Ctrl key while dragging the toolbar.
 - Drag a toolbar by the grip at its left (or top) edge () to undock.
 - To dock a toolbar vertically in *Linux*, the *Position* property of the toolbar must be set to *Left* or *Right*.

Closing a toolbar

1. In *Windows*: Place the cursor on a toolbar, then right click.
In *Linux*: Place the cursor on the menu bar, then right click.
A context menu displays.
2. Choose *BRICSCAD* on the context menu.
Toolbars that are open are checked.
3. In the toolbar list, uncheck the toolbar you want to close.

NOTE Floating toolbar can be closed by clicking the *Close* button ()

To set the icon size

You can choose between small (16 x 16) or large (32 x 32) icons on toolbars.

1. Move the cursor over a toolbar, then right click.
2. Choose *Toolbar size* then select either *Small Icons* or *Large Icons*.

To set the position and visible property of a toolbar

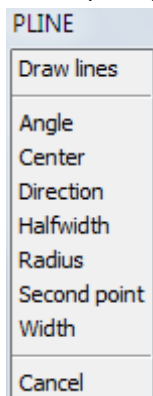
The *Visible* property determines whether a toolbar is visible when a cui-file is loaded.

The *Position* property determines where a toolbar displays when a cui-file is loaded.

1. In the Customize dialog click the *Toolbars* tab.
2. If necessary, expand the *BricsCAD* menu group.
3. Select the toolbar.
4. Choose *Position* in the *Properties* grid on the *Customize* dialog.
5. Click the settings field and choose the desired position in the list.
The options are: *Floating*, *Top*, *Left*, *Bottom*, *Right*.
6. Choose *Visible* in the *Properties* grid on the *Customize* dialog.
7. Click the settings field and choose either *Show* or *Hide* in the options list.
- 8.

Prompt Menus

BricsCAD commands often provide several options. These options appear in the status bar or command bar and also in a prompt menu, which initially displays in the top right corner of the screen. Choose the appropriate option by clicking in the prompt menu. E.g. click *Radius* in the *PLINE* prompt menu to define the radius of the current arc segment of a polyline.



To control the display of prompt menus


Whether a prompt menu displays and its location is controlled through the *PROMPTMENU* system variable.

Type *promptmenu* in the command bar, then type a value of 0 to 5 (see table below) and press Enter.

Value	Description
0	No prompt menus
1	Display prompt menu at user defined location (*)
2	Display prompt menu at top left corner
3	Display prompt menu at top right corner
4	Display prompt menu at bottom left corner
5	Display prompt menu at bottom right corner

(*) You can move the prompt menu by dragging it; future prompt menus will appear where you last placed it.

To edit the *PROMPTMENU* system variable in the Settings dialog:

- Do one of the following:
 - Click the *Settings* tool button () in the *Standard* toolbar.
 - Choose *Settings...* in the *Settings* menu.
 - Type *settings* in the command bar, then press Enter.

The *Settings* dialog opens.

2. Do one of the following:
 - Go to the *Prompt Menu* system variable under *Program Options > Display*.
 - Type *prompt menu* in the *search* field.
 - Click the down arrow at the right hand side of the *prompt menu* settings field, then select the option of your choice in the list.

NOTE The *PROMPTMENU* system variable is saved in the registry and therefore applies to all drawings.

Properties Bar

Command: STATBAR




The Properties Bar (Internet connection needed)

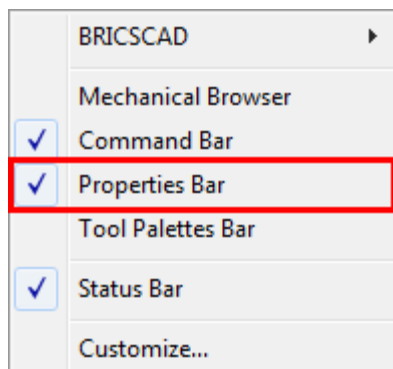
In the Properties Bar you can:

- Set the current properties: *color*, *layer*, *linetype*, *linetype scale* and *lineweight*
- Edit the properties of a single entity
- Edit the shared properties of a selection set
- Edit endpoints of lines and vertices of polylines graphically

To open the Properties Bar

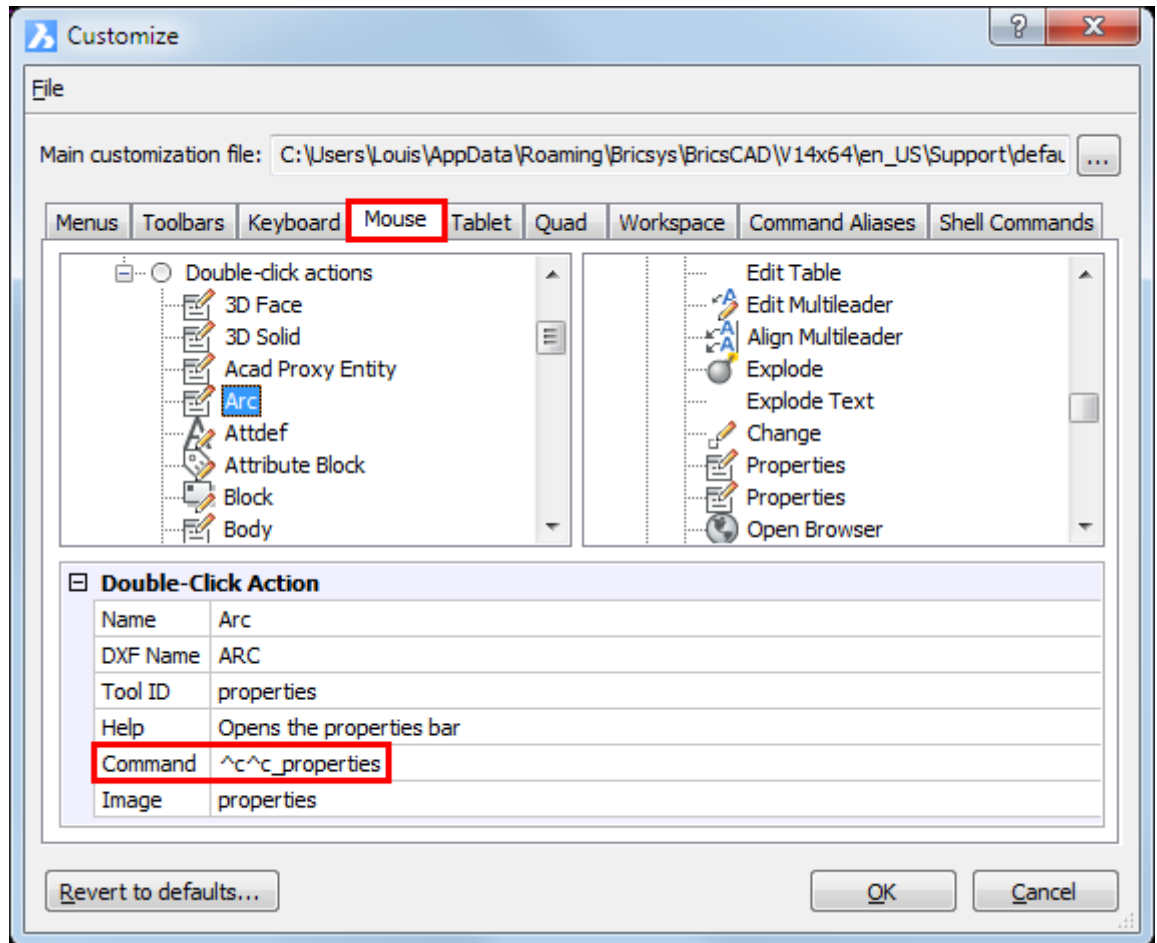
Do one of the following.

- Click the *Properties* tool button () on the Standard toolbar.
The pressed state of the tool button indicates that the Properties bar is currently open.
Click the tool button again to close the Properties bar.
- Right click when the cursor is on a toolbar.
A context menu displays.
The marked items in the context menu are currently open.
Select *Properties Bar* in the context menu.



Double click an entity.

The *Properties Bar* opens, showing the properties of the selected entity.
Double clicking entities such as texts, blocks or polylines start an editing command (See the *Mouse* tab on the *Customize* dialog).



- Click the *Properties* tool button (🔧) on the *Standard* toolbar.
- Choose *Properties* in the *Modify* menu.
- Type *properties* in the command window, then press Enter.

Properties Bar

No Selection

General	
Color	<input type="checkbox"/> ByLayer
Layer	0
Linetype	ByLayer
Linetype scale	1
Lineweight	ByLayer
View	
Camera	0.0, 0.0, 1.0
Target	0.0, 0.0, 0.0
Perspective	Off
Lens Length	50.0
Field of view	38.5801
Height	376.3
Width	780.3
Visual Style	2dWireframe
Misc	
Annotation scale	1:1

Properties Bar - No selection

The current entity properties, View properties and Annotation scale display

Properties Bar

Polyline

General	
Color	<input checked="" type="checkbox"/> ByLayer
Layer	FURNITURE
Linetype	ByLayer
Linetype scale	2.54
Plot style	ByColor
Lineweight	ByLayer
Hyperlink	
Thickness	0
3D Visualization	
Material	ByLayer
Geometry	
Vertices	4
Vertex	1
Position	1153.62, 871
X	1153.62
Y	871
Start width	0
End width	0
Bulge	0
Global width	0
Elevation	0
Area	3600
Length	240
Misc	
Closed	Yes
Linetype general	Disabled

Properties Bar - Single entity selected

The properties of the selected entity display

Properties Bar

Polyline (2)

General	
Color	<input checked="" type="checkbox"/> ByLayer
Layer	FURNITURE
Linetype	ByLayer
Linetype scale	2.54
Plot style	ByColor
Lineweight	ByLayer
Hyperlink	
Thickness	0
3D Visualization	
Material	ByLayer
Geometry	
Vertices	4
Global width	0
Elevation	0
Area	10200
Length	650
Misc	
Closed	Yes
Linetype general	Disabled

Properties Bar - Multiple entities selected

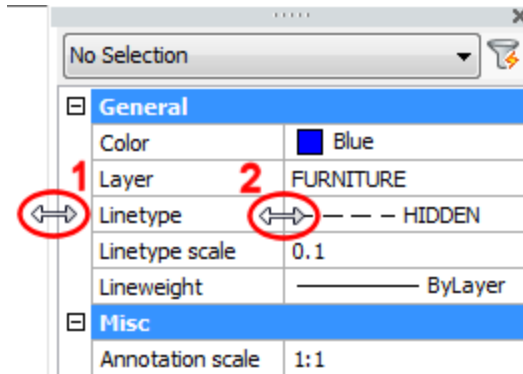
The shared properties of the selected entities display

NOTE The *Properties Bar* can be either floating or docked. To dock the *Properties Bar*, drag it by its title bar to either the left or right hand side of the BricsCAD application window.

NOTE Read-only fields display in grey. The content of a read-only field can be copied though. To copy a read-only field, click the field, then press Ctrl C.

To adjust the size of the Properties Bar

1. Move the cursor over one of the edges of the *Properties Bar* (1).
The cursor turns into a double-headed arrow.
2. Press and hold the left mouse button to drag the edge of the *Properties Bar*.



3. Move the cursor over the boundary between the Setting Names and the Setting Fields columns (2).
The cursor turns into a double-headed arrow.
4. Press and hold the left mouse button to drag the boundary.

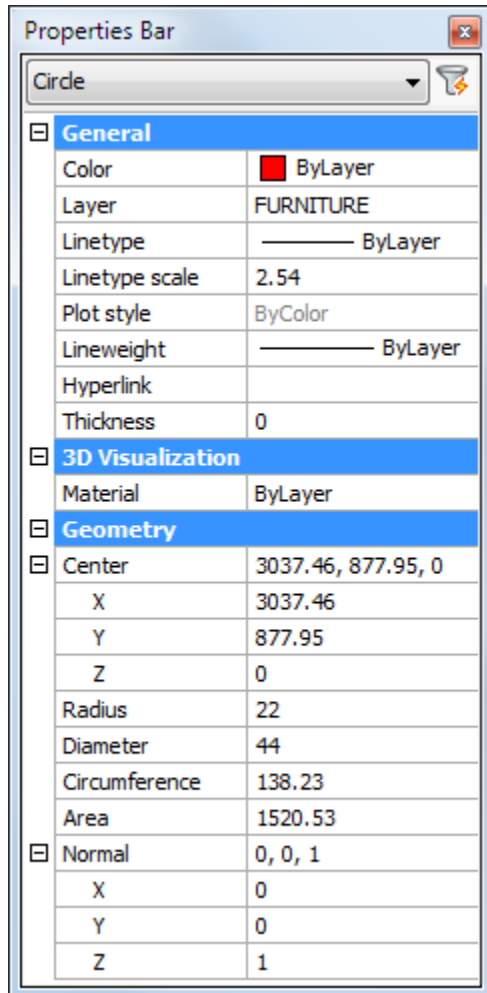
NOTE When docked, only the left (or right) edge of the *Properties Bar* is adjustable.

Setting the current properties in the Properties Bar

1. (option) Click on *Color*,
then click the down arrow button to select a color.
2. (option) Click on *Layer*,
then click the down arrow to select a layer.
3. (option) Click on *Linetype*.
then click on the down arrow to select a linetype.
4. (option) Click on *Linetype scale*,
then type the new value in the *Linetype scale* field.
5. (option) Click on *Lineweight*,
then click on the down arrow to select a lineweight.

Edit the properties of a single entity

1. Select the entity.
The entity's properties display in the *Properties Bar*.



Properties of a circle

2. Click the property you want to modify.
The settings field of the selected property is activated.
3. Type a new value in the settings field of the selected property or choose a setting from the list box, then press the *Enter* key or select another property.
The entity is updated.
4. (option) repeat steps 2 and 3 to modify another property.
5. Press the *Escape* key to stop.

NOTES


- Properties of which the value displays in grey cannot be modified.
- Step 3: Properties (such as the General properties), which are chosen from a list are updated instantly.

To edit the endpoints of a line

1. Select the line.
2. On the *Properties Bar*, under *Geometry*, do one of the following:
 - Select the X-, Y- or Z- field, then enter a value.
 - Select either *Start point* or *End point*.

General	
Color	<input type="checkbox"/> ByLayer
Layer	SEWER
Linetype	ByLayer
Linetype scale	2.54
Plot style	ByColor
Lineweight	ByLayer
Hyperlink	
Thickness	0
3D Visualization	
Material	ByLayer
Geometry	
Start point	3008.47, 850.43, 0
X	3008.47
Y	850.43
Z	0
End point	3106.4, 752.5, 0
X	3106.4
Y	752.5
Z	0
Delta	97.93, -97.93, 0
X	97.93
Y	-97.93
Z	0
Length	138.5
Angle	315°0'0"

An X indicates the point being edited.

- Click the *Pick Point* button ().
 - Specify the point in the drawing.
3. Press the *Escape* key to stop.


NOTE This procedure also applies to edit the insertion point of texts, images, blocks, Xrefs, ...

To edit the vertices of a polyline

1. Select the polyline..
2. On the *Properties Bar* under *Geometry*, select *Vertex*.
3. Click the *Next/Previous* arrow buttons to select a vertex.

General	
Color	■ ByLayer
Layer	FURNITURE
Linetype	ByLayer
Linetype scale	2.54
Plot style	ByColor
Lineweight	ByLayer
Hyperlink	
Thickness	0
3D Visualization	
Material	ByLayer
Geometry	
Vertices	7
Vertex	1 ◀ ▶
Position	3245.5, 759
X	3245.5
Y	759
Start width	0
End width	0
Bulge	-0.41421356
Global width	0
Elevation	0
Area	10054.78
Length	426.81
Misc	
Closed	Yes
Linetype generation	Disabled

An X indicates the vertex being edited.

4. Do one of the following:
 - Select the X- or Y- field, then enter a value.
 - Under *Vertex* select *Position* and click the *Pick Point* button () , then specify the point in the drawing.
5. Press the *Escape* key to stop.

Edit the shared properties of a selection set

1. Select the entities.
The shared properties display in the *Properties Bar*.
2. Click the property you want to modify.
The settings field of the selected property is activated.
3. Type a new value in settings field of the selected property or choose a setting from the list box, then press the *Enter* key or select another property.
All selected entities are updated simultaneously.
4. (option) Repeat steps 2 and 3 to modify another property.
5. Press the *Escape* key to stop.

NOTES

- *Varies* displays for shared properties which are defined differently. If you edit such property, all entities in the selection set will be equally defined for this property.
- Step 3: Properties (such as the General properties), which are chosen from a list are updated instantly.

Keyboard Shortcuts

The following keyboard shortcuts are available:

Keyboard	Description
F1	Opens the BricsCAD Help.
F2	Toggles the display of the Prompt History window.
F3	Toggles Entity Snaps.
F4	Toggles the tablet mode.
F5	Sets the isoplane (see Using Isometric snap).
F6	Toggles the Dynamic UCS feature.
F7	Toggles the display of the grid.
F8	Toggles the Ortho setting. Disables Polar Tracking, if on.
F9	Toggles the Snap setting.
F10	Toggles the Polar Tracking setting. Disables Ortho, if on.
F11	Toggles the Entity Snap Tracking setting.
F12	Toggles the Quad display.
Alt+F8	Launches the VBARUN command.
Alt+F11	Launches the VBAIDE command.
Ctrl+1 (*)	Toggles the properties bar.
Ctrl+2 (*)	Opens the Drawing Explorer.
Ctrl+9 (*)	Toggles the display of the command bar.
Ctrl+A	Selects entities.
Ctrl+B	Toggles the Snap setting.
Ctrl+C	Copies the selection to the clipboard (COPYCLIP command).
Ctrl+Shift+C	Copies the selection along with a base point (COPYBASE command).
Ctrl+E	Sets the isoplane (see Using Isometric snap).
Ctrl+F	Toggles entity snaps on/off.
Ctrl+G	Toggles the display of the grid.
Ctrl+H	Toggles the value of the PICKSTYLE system variable, which controls the selection of groups and associative hatches.
Ctrl+I	Sets the readout of the coordinate field in the status bar.
Ctrl+J	Repeats the previously issued command.
Ctrl+K	Attach a hyperlink to an object or modify an existing hyperlink (HYPERLINK command).
Ctrl+L	Toggles the Ortho setting.
Ctrl+M	Repeats the previously issued command.
Ctrl+N	Creates a new drawing (NEW command).
Ctrl+O	Opens an existing drawing (OPEN command).
Ctrl+P	Prints the current drawing (PRINT command).
Ctrl+Shift+P	Toggles the properties bar.
Ctrl+Q	Quits the application; prompts to save changes (QUIT command).
Ctrl+R	Iterates through viewports.
Ctrl+S	Saves the current drawing (QSAVE command).
Ctrl+T	Turns the tablet on/off.
Ctrl+V	Pastes the contents of the Clipboard into the drawing (PASTECLIP command).
Ctrl+Alt+V	Pastes the contents of the Clipboard in a specified format (PASTESPEC command).
Ctrl+Shift+V	Pastes the contents of the Clipboard as a block (PASTEBLOCK command).

Ctrl+X	Cuts the selection and puts it on the clipboard (CUTCLIP command).
Ctrl+Y	Redoes the last action undone (REDO command).
Ctrl+Z	Undoes the last action (UNDO command).
Ctrl+[Cancels the running command.
Ctrl+\	Cancels the running command.
Shift+F2	Toggles the display of the command bar.
Shift+F3	Turns the Status Bar on/off (STATBAR command).
Shift+F4	Turns the Scroll Bars on/off (SCROLLBAR command).
Shift+F8	Launches the Visual Basic Project Manager (VBAMAN command).
Shift+F11	Launches the VBA COM Add-In Manager (ADDINMAN command).
PgUp	Moves the view up.
PgDn	Moves the view down.
Shift+Left Arrow	Moves the view to the left.
Shift+Right Arrow	Moves the view to the right.
Shift+Up Arrow	Moves the view up.
Shift+Down Arrow	Moves the view down.

(*) Do not press the Shift key on AZERTY-keyboards.

Model space and paper space

Understanding paper space and model space

When you start a drawing session, your initial working area is called *Model Space*. Model Space is an area in which you create two-dimensional and three-dimensional entities based on either the World Coordinate System (WCS) or a user coordinate system (UCS). You view and work in model space while using the *Model* tab.

In general model space consists of a single view that fills the screen. If needed, you can create additional views, called viewports, which can show different views of your drawing or 3D model. All viewports are displayed in a tiled manner. You can work in only one of these viewports at a time but all viewports are updated simultaneously. Click in a viewport to make it the current viewport. You can print the current viewport only. The Viewports command lets you manage your viewports in model space.

BricsCAD provides an additional work area, called *Paper Space*. Paper space represents a paper layout of your drawing. In this work area, you can create and arrange different views of your drawing similar to the way you arrange drawings on a sheet of paper. In paper space you can also add keynotes, annotations, borders, title blocks, and other print-related entities, which you don't want to see in model space.

Each drawing has at least one *Layout* in which you can have one or more *Layout Viewports*. Such layout viewports are to be considered as a view window on your drawing in model space. Layout viewports can be placed anywhere in a paperspace layout. Each viewport has its own scale and layer visibility. All viewports in the same layout can be printed simultaneously.

The Mview command lets you manage your viewports in paper space.

Although not necessary to print your drawing, paper space offers a lot of advantages:

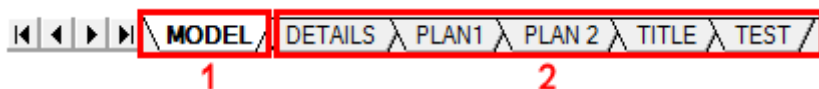
- Create multiple layouts to print the same drawing with different print settings, such as pen widths, printer configuration files, lineweight settings, drawing scale, and more.
- Add print-related entities that are not essential to the model itself, such as keynotes, annotations, title blocks, etc.
- For a single layout, create multiple layout viewports to print multiple views of your drawing at different scales.

Switching between model space and paper space

- To switch between Model Space and Paper Space Layouts use the Model and Layout tabs at the bottom of the drawing window.
- When working in a layout, use the MSPACE and PSPACE commands to toggle between paper space and model space.

Using the model and layout tabs

6. (option) To open model space, click the *Model* tab (1) at the bottom of the drawing window.
7. (option) To open a paper space layout, click the corresponding *Layout* tab (2) at the bottom of the drawing window.



NOTE A drawing contains at least one layout, which is named *Layout1* by default.

Toggle between model space and paper space in a layout

Commands: MSPACE and PSPACE

1. (option) When in *model space* (Model Space, with floating viewports) do one of the following to switch to *paper space*:
 - Type either *pspace* or *ps*, then press Enter
 - Double click outside a viewport.
2. (option) When in *paper space* do one of the following to switch to *model space* (Model Space, with floating viewports):
 - Type either *mspace* or *ms*, then press Enter.
 - Double click inside a viewport.

NOTE *MSPACE* and *PSPACE* commands are available when working in a layout only.

Customizing BricsCAD



Customize BricsCAD (Internet connection needed)

There are several ways to customize BricsCAD. You can change the look of BricsCAD by adapting the layout of menus and toolbars but you can also compose macro's, write LISP routines or use VBA to change the way BricsCAD works.

All customization that affects the user interface is done in the Customize dialog and saved in CUI files. CUI stands for **C**ustomize **U**ser **I**nterface. The definitions for all menus, toolbars and keyboard shortcuts are loaded from the main CUI file (default.cui) when you launch BricsCAD.

Additional definitions, such as your own customizations, are loaded from partial CUI files.

Aliases and shell commands are loaded from the *default.pgp* file.

NOTE It is recommended not to modify the default.cui file. With each new update, the BricsCAD installer replaces the existing default.cui. If the existing default customization file has changed, you will be prompted whether or not keep the existing file (see the BricsCAD user file manager). However, if new tools are added in an update, you will not be able to access them until you load the new default.cui. If you are not sure whether the most recent default customization file is used, press the *Revert to default* button on the Customize dialog.

Store your customizations in a partial CUI file, with each new update the default.cui is updated automatically. To restore your customizations, reload the partial CUI file. The reference to your partial CUI file is stored in the main cui file (default.cui).

In any case if you customize menus, toolbars, accelerators, aliases, it is recommended to keep a copy of these files, including default.cui and default.pgp, before doing an update.

Another approach to keeping your menu customization when changing to a new version of BricsCAD is to make your custom CUI file the "main" CUI file (see: To load a main cui file), and attach "default.cui" to it as a "partial" CUI file (see: To load a partial cui file). Then you don't have to do anything when you update. The new "default.cui" overwrites the old one, and replaces it as the "partial" CUI attached to your custom CUI file. It automatically updates, very much like an xref drawing file that you edit or replace with another file of the same name. Another advantage of this method is that if you've customized the context menus, your versions are used instead of the default ones, since yours are in the "main" file. You don't have to delete the default context menus, or make any change at all to the new default menu file when you update.

In any case if you customize menus, toolbars, accelerators, aliases, it is recommended to keep a copy of these files, including default.cui and default.pgp, before doing an update.

BricsCAD Startup Options

When starting up BricsCAD it is possible to add arguments or switches. E.g. the command line "BricsCAD.exe drawing1.dwg drawing2.dwg drawing3.dwg" will start the BricsCAD application and load the specified drawings while the command line "bricscad.exe /P MyProfile" will load BricsCAD using user profile MyProfile.

"/P MyProfile" is called a command line switch. Both a slash or a hyphen are accepted to precede the switch, e.g. "-P MyProfile" will also work. The space is optional, e.g. "/PMyProfile" would also work, and the switch character is case insensitive, e.g. "-p MyProfile" is also fine.

Switch	Description
/P [userProfileName]	Starting without the /P option, BricsCAD re-uses the last used user profile (called current user profile), which is not necessarily the Default profile. If the user profile specified with the /P option doesn't exist at startup, a new empty user profile will be created with that name. If the /P argument is an .arg file the arg file is used to initialize the profile settings on the first run (See Starting BricsCAD with a specific profile).
/B [scriptFileName.scr]	Runs a script at startup.
/L	Suppresses the startup splash screen.
/LD [application.arx/brx/drx]	Loads an application.
/S [path]	Redefines the SRCHPATH system variable. e.g. BricsCAD.exe /S C:\BricsCAD\
/T [TemplateFileName]	Creates a new drawing based on the template drawing specified.
regserver unregserver	To register/unregister BricsCAD COM. Please note these arguments are to be used without a preceding "/".

- NOTES**
- Switch arguments containing spaces should be enclosed by double quotes, e.g. /P "My user profile"
 - Switches can be combined, e.g. /P MyProfile /L /S C:\BricsCAD\ will start BricsCAD using the MyProfile user profile without the splash screen and sets the SRCHPATH system variable to C:\BricsCAD\.

Besides these switches, BricsCAD supports file names as command line arguments.

Supported file types are:

DWG - to load a drawing

DXF - to load a drawing

SCR - to load a script

LSP - to load a LISP program

DLL - to load an ADS/SDS or DRX/ARX program

SLD - to load a slide file

MNU - to load a menu file

CUI - to load a menu file

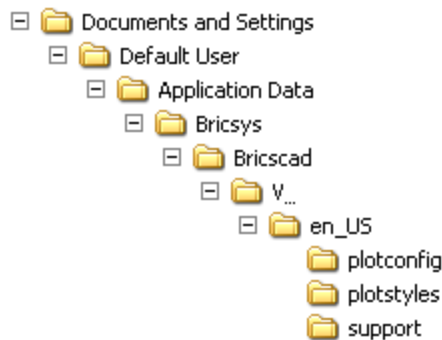
BricsCAD user files

In BricsCAD the preferences of each user are stored in specific folders, which are called the *Local root* folder and the *Roamable root* folder.

The *Local root* folder contains the *Template* folder, where the drawing templates are saved.

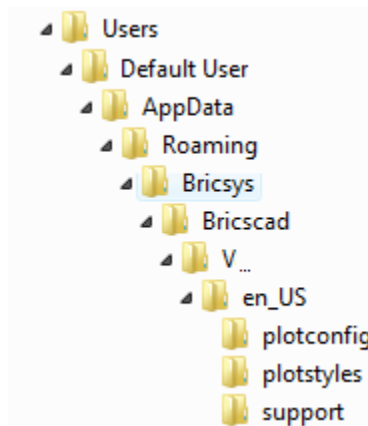
The *Roamable root* folder contains three subfolders:

- *plotconfig*: where the plotter configuration files (*.pc3) are saved
- *plotstyles*: where the plot style table files (*.stb) and plot color table files (*.ctb) are saved
- *support*: where the hatch pattern files (*.pat), line style files (*.lin), alias files (*.pgp), unit files (*.unt), custom user interface files (*.cui), etc. are saved.



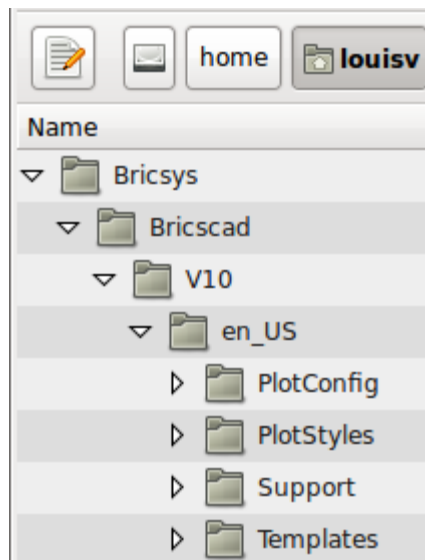
Roamable root folder in Windows XP

C:\Documents and Settings\<user name>\Application Data\Bricsys\BricsCAD\V...\en_US



Roamable root folder in Windows Vista or higher

C:\Users\<user name>\AppData\Roaming\Bricsys\BricsCAD\V...\en_US



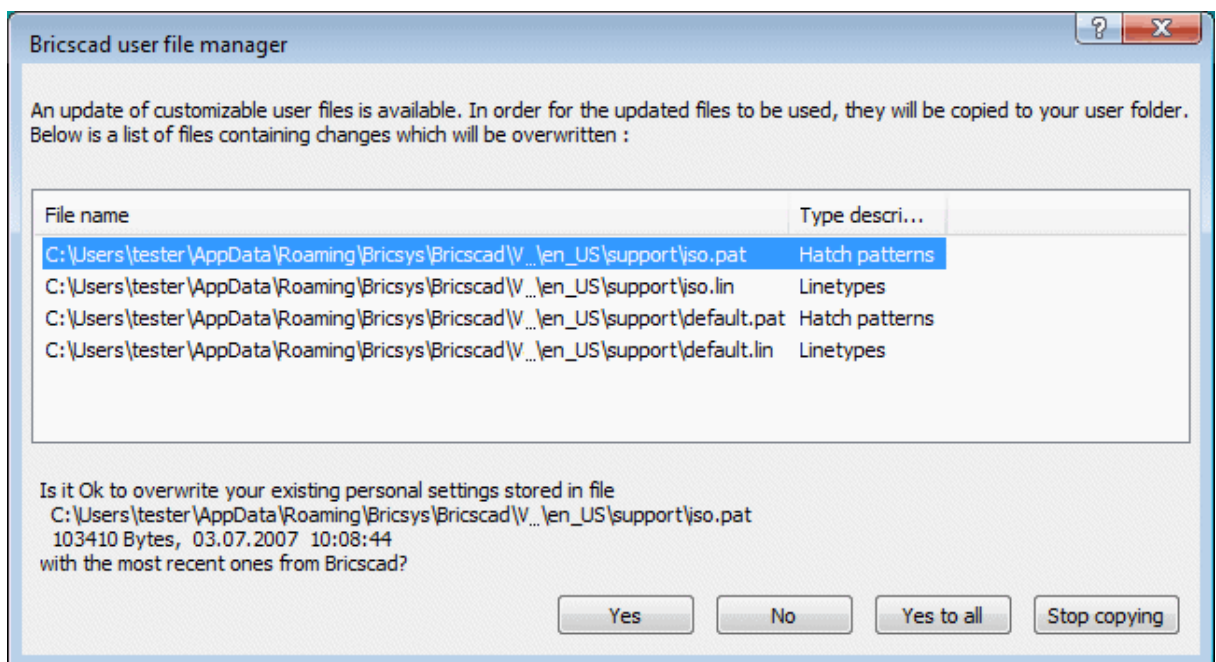
Roamable root folder in Linux

/home/<user name>/Bricsys/BricsCAD/V.../en_US

The BricsCAD user file manager

When you install an update of the software, the first time you launch BricsCAD after the update, the content of the *User Data Cache* folder (C:\Program Files\Bricsys\BricsCAD\UserDataCache) is compared with the content of the *Local root* folder and the *Roamable root* folder of the current user. If you have customized the files in these folders and if one or more of the corresponding files in the *User Data Cache* folder are more recent, the *BricsCAD user file manager* is launched asking you whether to keep your files or to overwrite your files with the updated files.

1. Do one of the following:
 - Select a file in the file list, then click the *Yes* button to overwrite or click the *No* button to keep your file.
Repeat this procedure for each file in the list.
 - Click the *Yes to all* button to replace all your files with the more recent files from BricsCAD.
 - Click the *Stop Copying* button to keep all your files.



2. If you have decided not to update one or more files, an alert window displays. Do one of the following:
 - Click the *Yes* button if you want to update one of these files later.
In this case this procedure is restarted the next time you start BricsCAD.
 - Click the *No* button if you don't want to be reminded.

Understanding the Customize Dialog

What are the CUI files

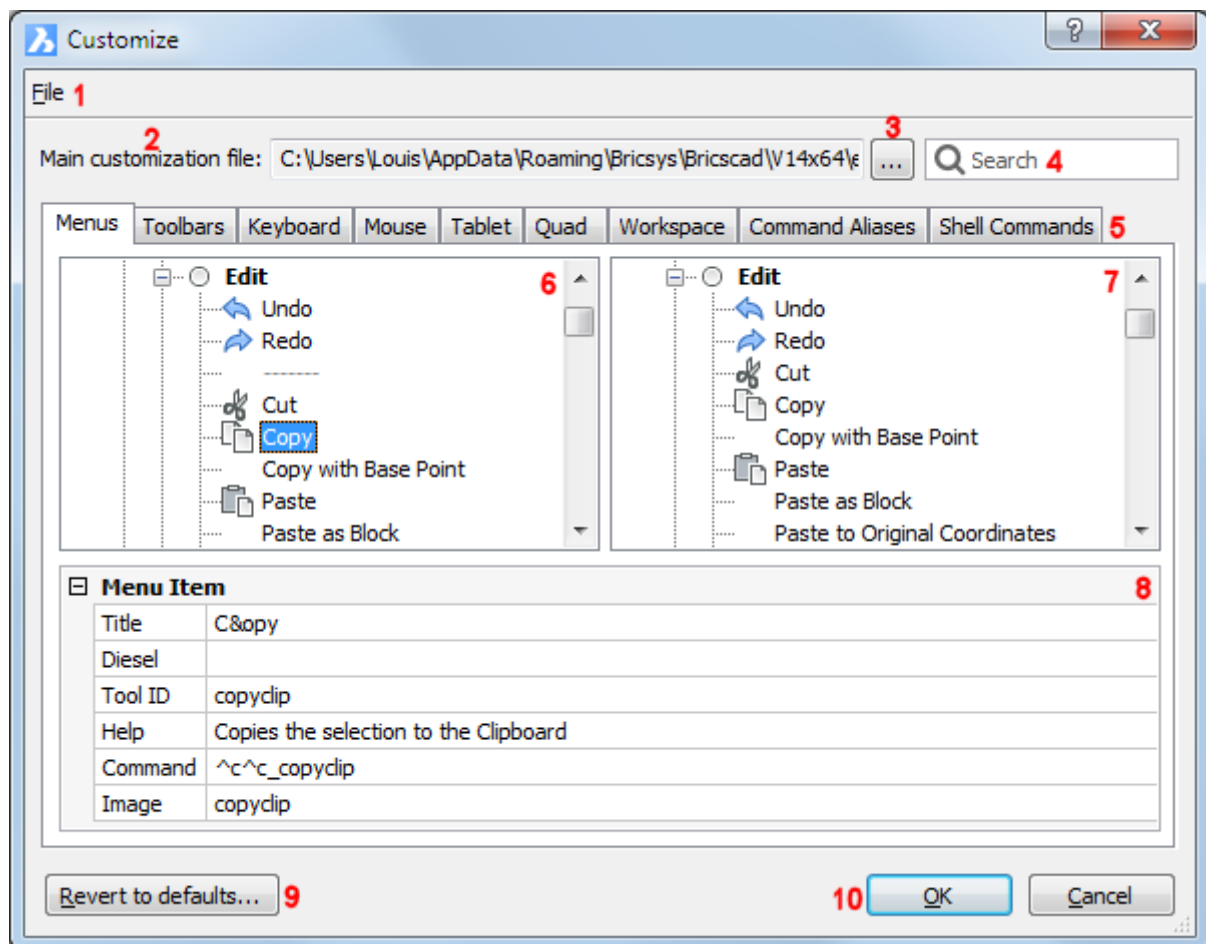
Files with the *CUI* extension are **C**ustomize **U**ser **I**nterface files for BricsCAD. These XML format files replace the previously used menu definition files (*.mnu *.mns). CUI files can be edited with the build-in interactive editor, launched by the CUSTOMIZE command. You can migrate CUI files between BricsCAD versions and combine them between different computers.

NOTE The Load partial CUI file ... procedure accepts IntelliCAD menu files (*.mnu *.mns) and converts them to CUI files.

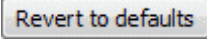
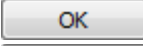
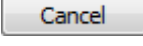
Opening the Customize dialog

To open the *Customize* dialog do one of the following:

- Choose *Customize* in the *Tools* menu
- In *Windows*: Hover over a toolbar or a blank portion of the toolbar area, then right click and choose *Customize* in the context menu.
- In *Linux*: Hover over a blank portion of the menu bar, status bar or toolbar area, then right click and choose *Customize* in the context menu.
- Type *customize* in the command window, then press Enter.
- Type *cui* in the command window, then press Enter.



The components of the *Customize* dialog are:

1. *File* menu
2. Main customization file name and location
3. Browse for main customization file button
4. Search field: type a search string then click the looking glass icon, click again for the next hit.
5. Section tabs
6. Content of the currently selected section
7. *Available tools* tree
8. Properties grid, displaying the properties of the selected item
9.  Reloads default menus, toolbars, etc. overwriting the customized *default.cui* file.
10.  Saves changes and closes the *Customize* dialog.
 Closes the *Customize* dialog, without saving changes.

Loading and unloading CUI files

Commands: MENULOAD, CUILOAD, MENUUNLOAD and CUIUNLOAD


When you install BricsCAD the *default.cui*, *compact.cui* and *bonus.cui* files are copied to the *Support* folder of your Roamable root folder. When you launch BricsCAD the first time the *default.cui* file is loaded automatically.

There are two procedures to load a CUI file:

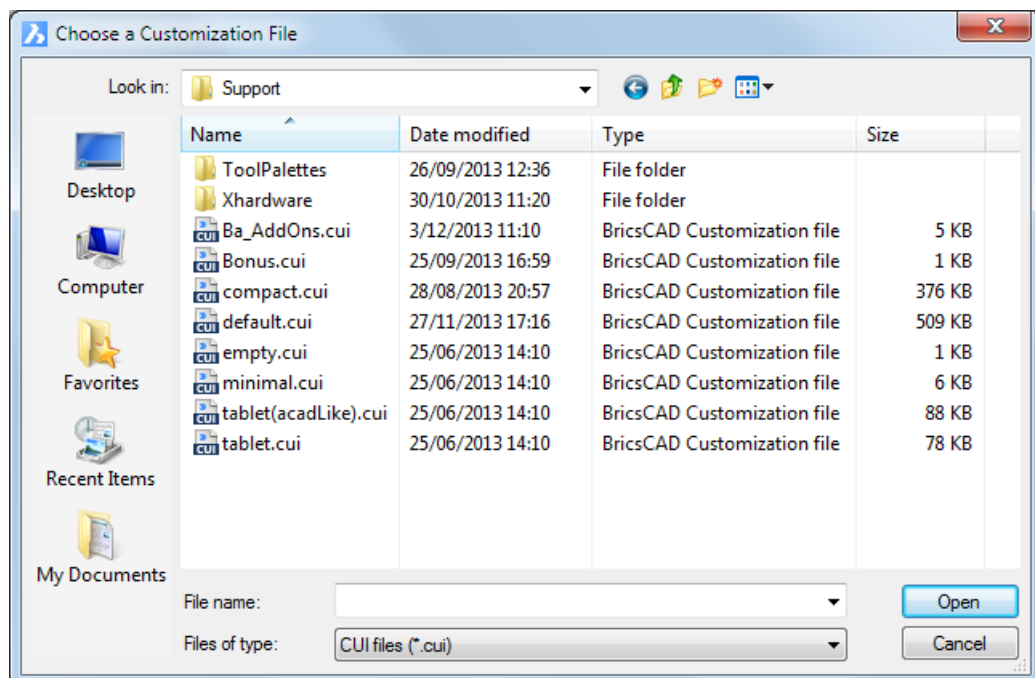
- Load main CUI file, which replaces all existing content with the content of the new CUI file.
- Load partial CUI file, which appends its content to the content of the currently loaded main CUI file.

NOTE You can load an IntelliCAD menu file if you select *MNU files (*.mnu;*.mns)* in the *Files of Type* list box of the *Choose a Customization file* dialog in the Load main CUI file, Load partial CUI file and Manage Customization Groups procedures. The menu file will be converted to a CUI file automatically.

To load a main CUI file

1. Do one of the following:
 - Click the *Load Menu* tool button () on the *Tools* toolbar.
 - Choose *Load Menu* in the *Tools* menu.
 - Type *menu* in the command bar, then press Enter.
 - Open the Customize dialog, then click the *Browse* button.
 - Open the Customize dialog, then choose *Load main CUI file ...* in the *File* menu.

The *Choose a Customization file* dialog displays.



2. Do one of the following:
 - Double click a *CUI* file.
 - Select a *CUI* file, then click the *Open* button.

The selected *CUI* file is loaded.

3. Close the *Customize* dialog.

To load a partial CUI file

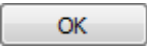
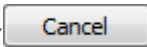
1. Open the *Customize* dialog, then choose *Load partial CUI file ...* in the *File* menu.

The *Choose a Customization file* dialog displays.

2. Do one of the following:
 - Double click a CUI file.
 - Select a CUI file, then click the *Open* button.

The partial CUI file is loaded.

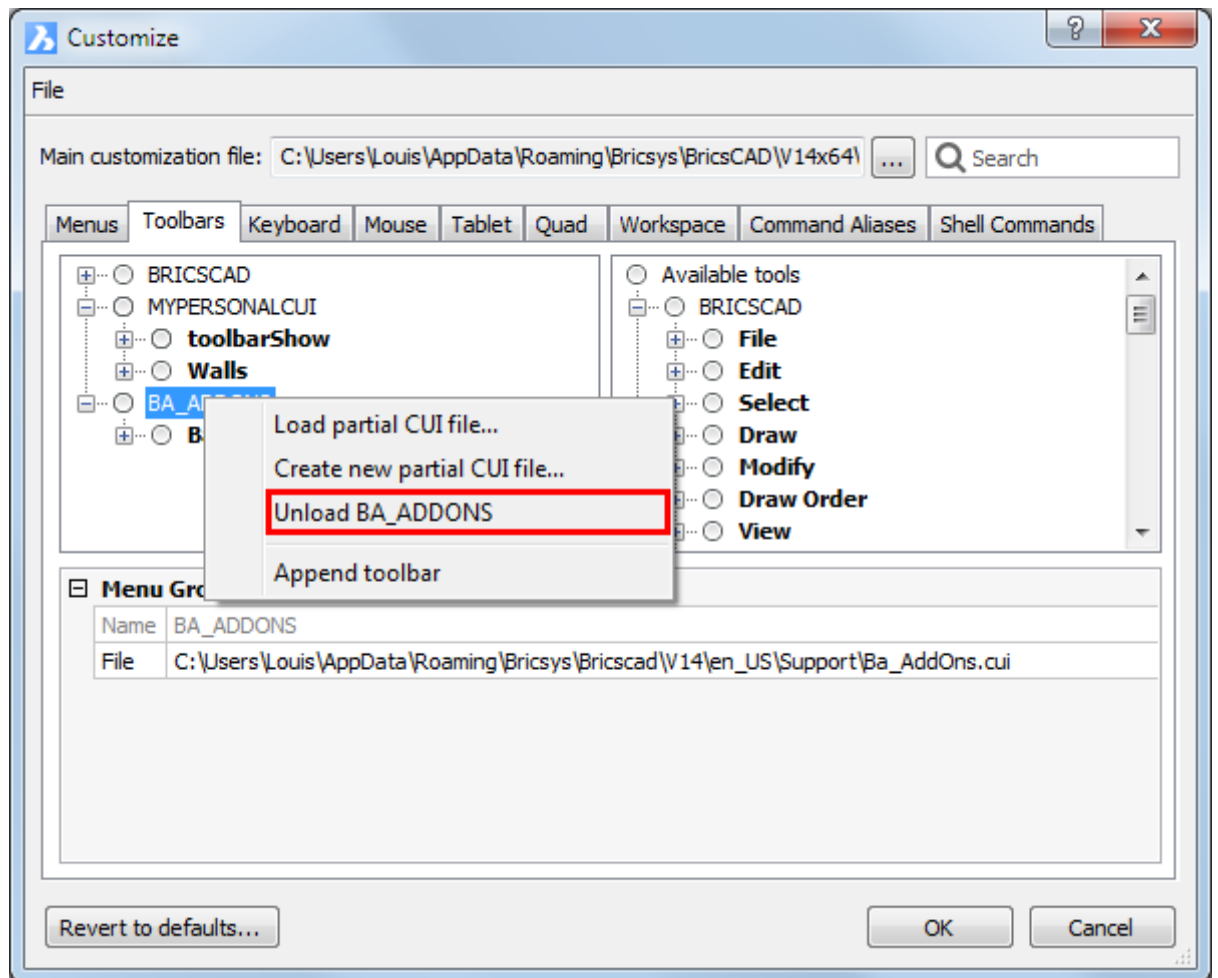
3. (option) Repeat steps 1 and 2 to load another partial CUI file.

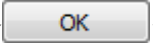
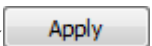
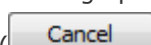
4. Do one of the following:
 - Click the *OK* button () to confirm the loading of the partial CUI file(s) and close the *Customize* dialog.
 - Click the *Cancel* button () close the *Customize* dialog and unload the partial CUI file(s).

NOTE If a CUIX file is selected in step 2 it will be converted to a CUI file on-the-fly. When a CUIX file contains bitmaps, they are copied into a .RESZ resource file, which are supported as an alternative to resource DLL (zip file containing tool bitmaps; works on all platforms). The following image formats are supported: BMP, JPG, PNG, ICO, GIF, TIFF, XBM, XPM.

To unload a partial CUI file

1. Open the *Customize* dialog.
2. In the *Menus*, *Toolbars* or *Keyboard* section select the partial CUI file you want to unload.

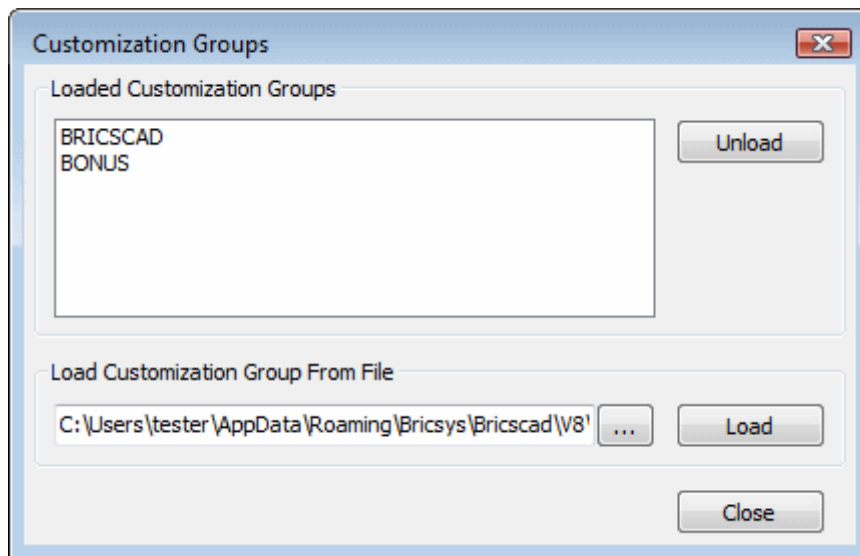



3. Right click and select *Unload <selected partial CUI>* in the context menu.
4. Do one of the following:
 - Click the *OK* button () to confirm the unloading of the partial CUI file and close the *Customize* dialog.
 - Click the *Apply* button () to confirm the unloading of the partial CUI file and keep the *Customize* dialog open.
 - Click the *Cancel* button to () close the *Customize* dialog and reload the partial CUI file.

Managing customization groups

1. Type *Menuload* in the command bar, then press Enter.

2. The *Customization groups* dialog opens.



3. (option) To load a partial CUI file, click the browse button () to select a partial CUI file, then click the *Load* button.
4. (option) To unload a partial CUI file, select the partial CUI file in the *Loaded Customization Groups* list, then click the *Unload* button.
5. Click the *Close* button to stop.

•

Creating custom tools

General procedure to create custom tools

1. Create a Partial CUI file.
2. (option) Add toolbox(es) in a Menu Group.
3. (option) Add tools in a toolbox
4. (option) Add main menu(s)
5. (option) Add sub menu(s)
6. (option) Add context menu(s)
7. (option) Add toolbar(s)
8. (option) Add flyout(s) to toolbar(s)
9. (option) Add control(s) to toolbar(s)
10. (option) Add keyboard shortcuts
11. (option) Add existing tools to menus, toolbars or shortcuts
12. (option) Create new tools in menus, toolbars or shortcuts
13. (option) Change the tool order in menus and toolbars

To create a Partial CUI file

1. In the Customize dialog do one of the following:
 - In the *File* menu, choose *Create new partial CUI file ...*
 - Place the cursor on Menu group, then right click and choose *Create new partial CUI file ...* in the context menu.

The *Create a Customization File* dialog opens.

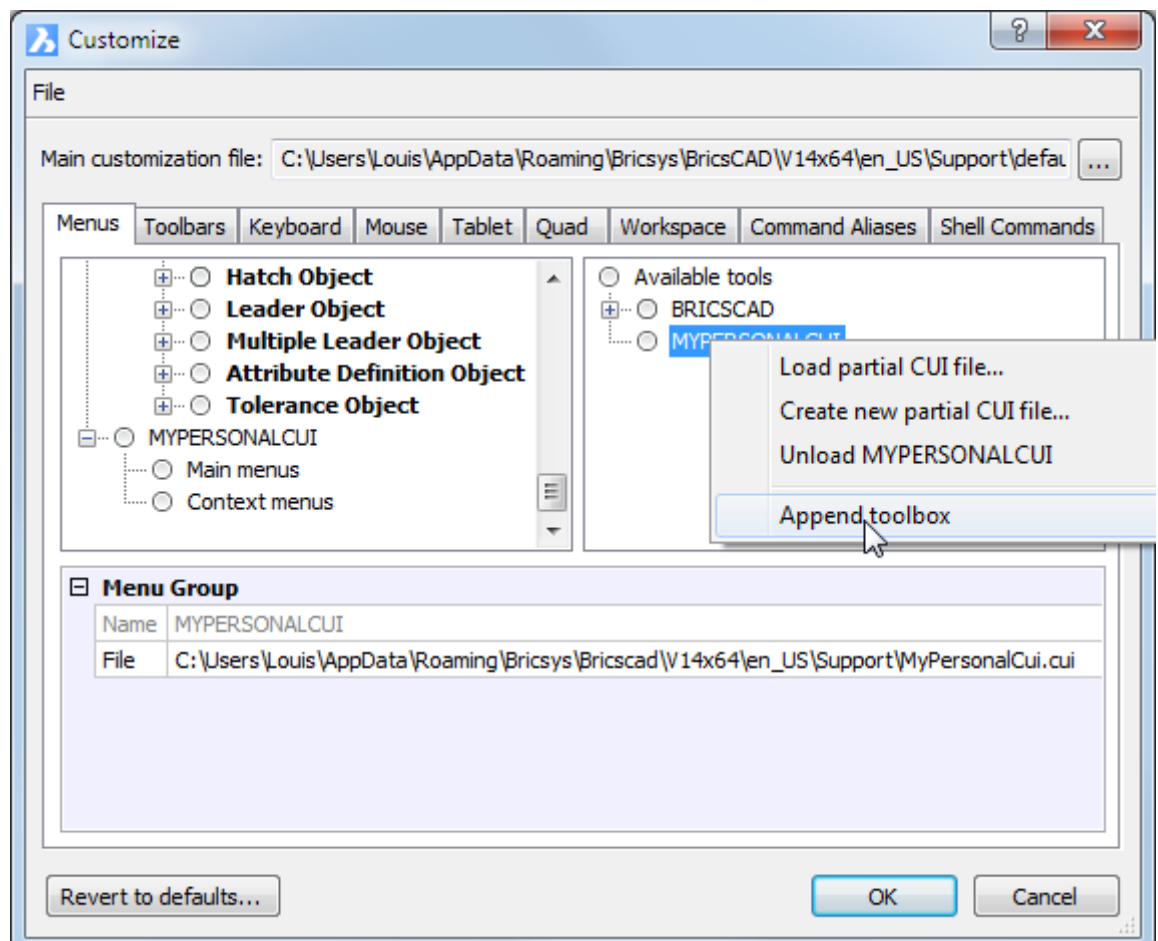
By default the new partial CUI file will be created in the *Support* folder of the Roamable root folder.

2. (option) Select a different folder to save the CUI file.
3. Type a name for the new CUI file in the *File name* field.
4. Click the *Save* button.

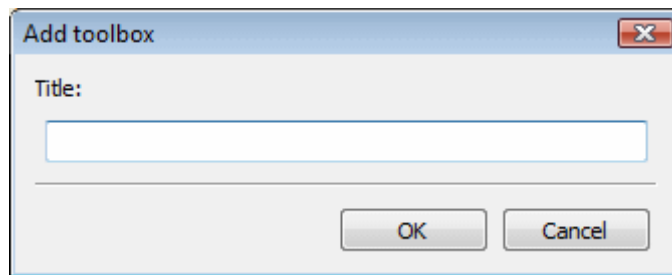
The newly created *Partial CUI file* is loaded and a *Menu Group* of the same name is created in the *Menus* tree, *Toolbars* tree, *Keyboards* tree and *Available Tools* tree.

To add a toolbox in a menu group

1. In the Customize dialog select a menu group.
2. Right click and choose *Append toolbox* in the context menu.



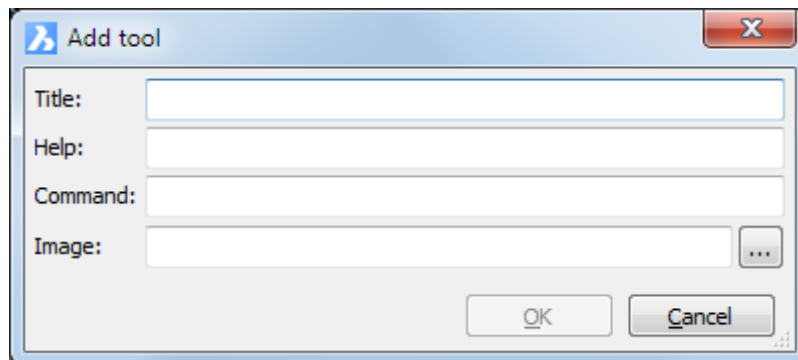
3. Type a name for the new *toolbox* in the *Title* field of the *Add toolbox* dialog.



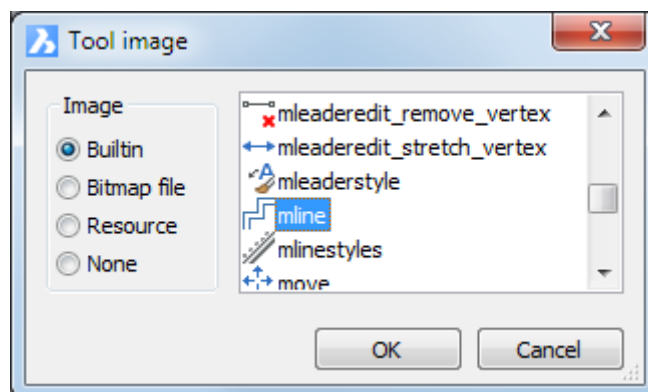
4. Click the *OK* button.
The toolbox is added in the *Available Tools* tree.
5. Click the *Apply* button to save the definition.

To add a tool in a toolbox

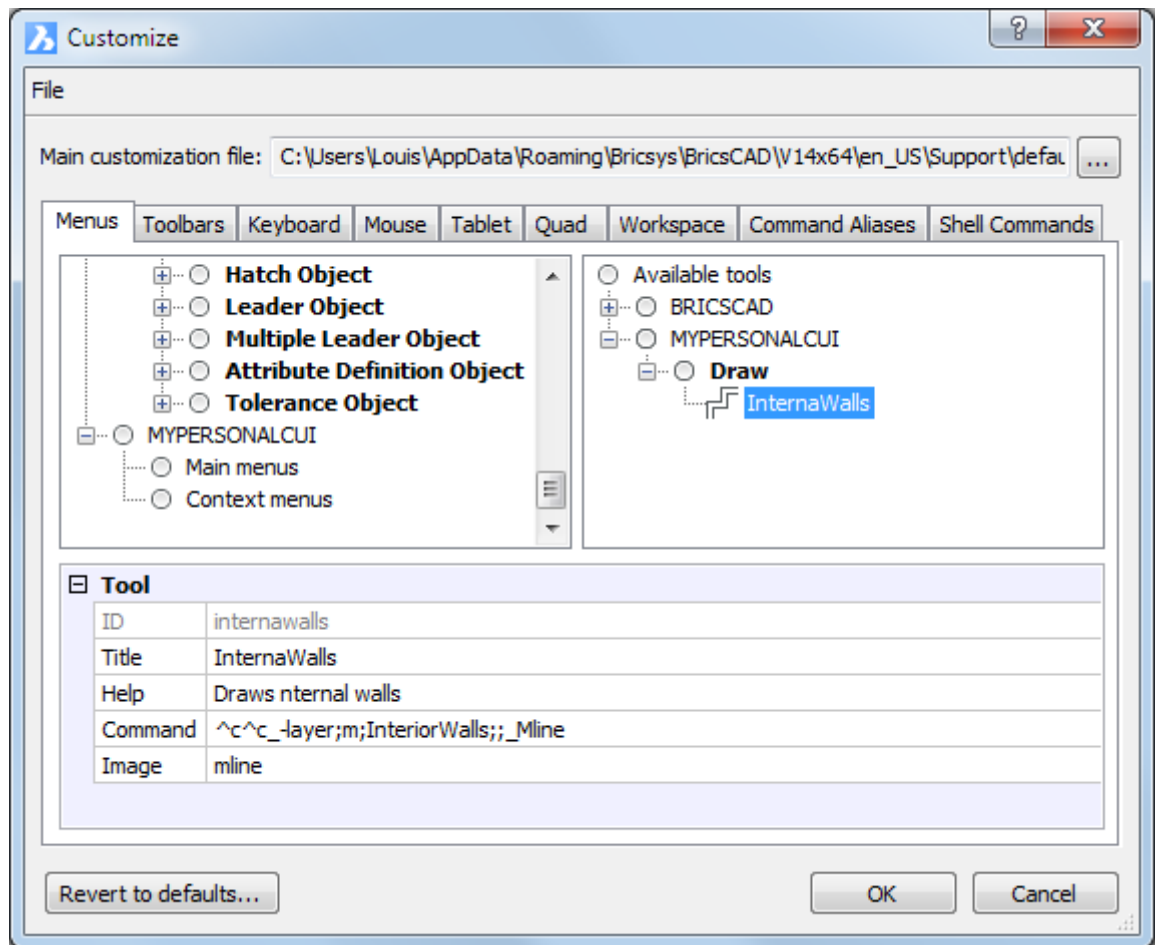
1. In the *Customize* dialog select a toolbox in the *Available Tools* tree.
2. Right click and choose *Append tool* in the context menu.
The *Add tool* dialog displays.



3. Type a name for the new tool in the *Title* field of the *Add tool* dialog.
4. (option) Type a help string in the *Help* field of the properties grid.
5. Type a command or command macro in the *Command* field of the properties grid.
6. Click the *Browse* button (⋮), then select an image for the new tool.
Do one of the following:
 - Select *Builtin*, then choose one of the built-in icons.



- Select *Bitmap file*, and choose a bmp, gif, jpg, jpeg or png image file.
7. Click the *OK* button.
The tool is added in the toolbox.



In the above example the *Command* field contains a macro that makes the *InteriorWalls* layer current, then launches the *Mline* command. If the layer does not exist yet, it is created first.

NOTE

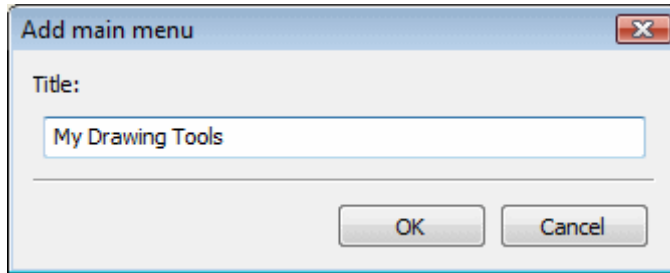
- ^c^c is the equivalent of pressing the Esc key twice, which first ends a running command if necessary.
- an underscore (_) in front of a command name ensures that the macro still works in a non-English version of BRICSCAD.
- a point (.) in front of a command name ensures that the macro still works if this command is temporarily undefined (see the UNDEFINE and REDEFINE commands).
- a minus sign (-) in front of a command launches the command line version of a command, thus preventing the command dialog window to open.
- a semicolon (;) is the equivalent of pressing the Enter key
- a backslash (\) waits for user input, e.g. specifying a point.

To add a main menu

1. In the Customize dialog click the *Menus* tab.
2. Do one of the following:
 - Click the *Main Menus* subgroup of the menu group, then right click and choose *Append main menu* in the context menu.
 - Select an existing menu in the *Main Menus* subgroup, then right click and choose *Insert main menu* in the context menu.

New menus are added above the selected existing menu or as the first menu if you select the *Main Menus* subgroup.

3. Type a name in the *Title* field of the *Add main menu* dialog.



4. Click the *OK* button.
5. (option) Repeat steps 2 through 5 to add more menus.
6. Click the *Apply* button to save the definition(s).

NOTE The new main menu is not added to the Menu bar until at least one menu items is added to it.

To add a submenu to a menu

1. In the Customize dialog click the *Menus* tab.
2. If collapsed, expand the menu group (e.g. BRICSCAD).
3. If collapsed, expand the *Main menus* group of the menu group.
4. Do one of the following:
 - Select the menu, then right click and choose *Append submenu* in the context menu.
 - Select an existing item in the menu, then right click and choose *Insert submenu* in the context menu.

New submenus are added above the selected existing item or as the first item if you select the menu.

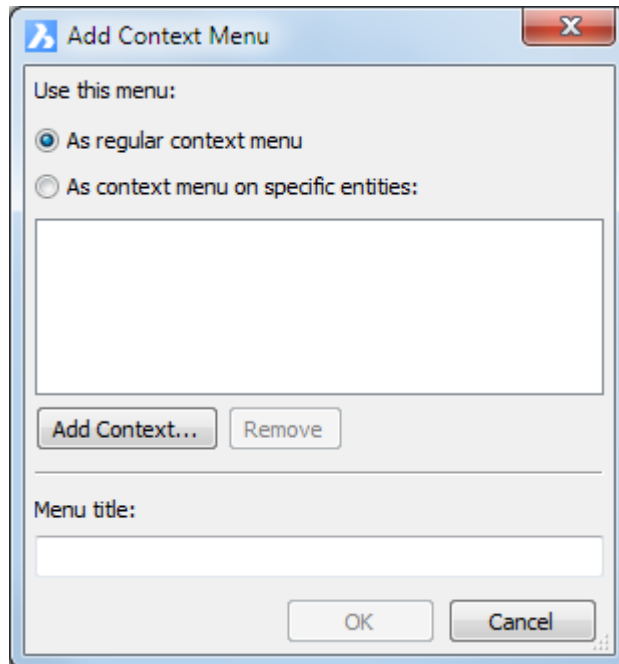
5. Type a name in the *Title* field of the *Add submenu* dialog.
6. Click the *OK* button.

To add a context menu

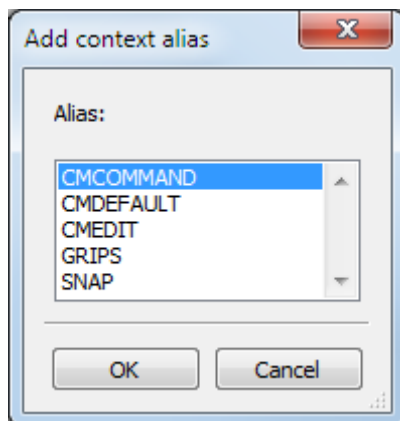
1. In the Customize dialog click the *Menus* tab.
2. Do one of the following:
 - Select the *Context Menus* subgroup of the menu group, then right click and choose *Append context menu* in the context menu.
 - Select an existing context menu, then right click and choose *Insert context menu* in the context menu.

New context menus are added above the selected existing context menu or as the first context menu if you select the *Context Menus* subgroup.

3. Select a menu in the *menu* list of the *Add context menu* dialog.



4. (option) Select *As regular context menu*, then click the *Add Context...* button. The *Add context alias* dialog box displays:



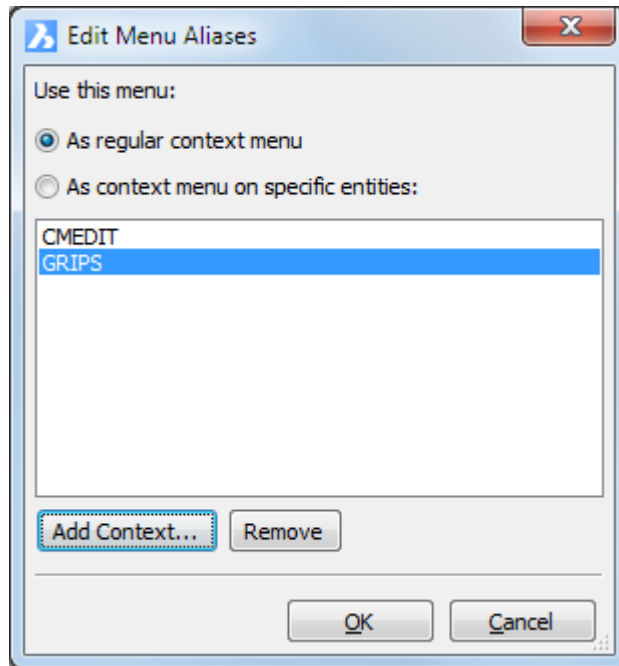
Choose:

- CMCOMMAND: to replace the default *Command* context menu.
- CMDEFAULT: to replace the default *Default* context menu.
- CMEDIT: to replace the default *Edit* context menu.
- GRIPS: to replace the default *Grips* context menu.
- SNAP: to replace the default *Entity Snap* context menu.

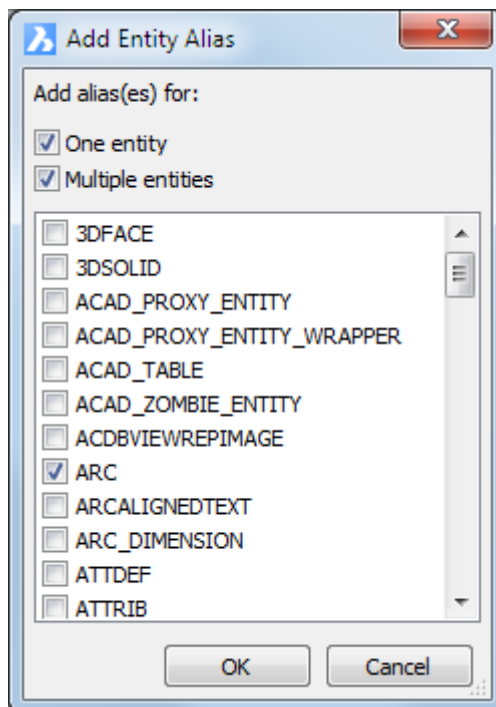
Click the *OK* button to confirm.

Optionally repeat the previous step to add another alias.

Select an alias, then click the *Remove* button to remove the alias.



5. (option) Select *As context menu on specific entities:*, then click the *Add Entity Type(s)...* button.
The *Add Entity Alias* dialog box displays:



Do one of the following:

- Check the *One entity* option: the context menu applies if one entity is selected only..
- Check *Multiple entities*: the context menu applies in multiple entities are selected only.
- Check both options: the context menu applies if one or more entities are selected.

Select one or more entity types in the list.

Click the *OK* button to confirm.

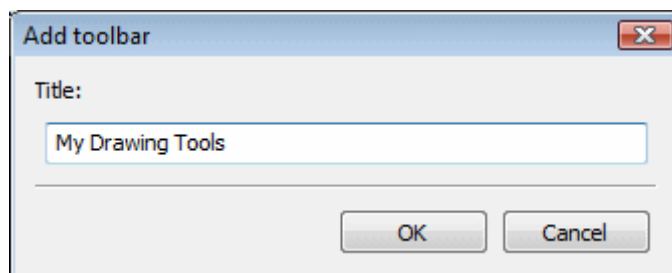
6. Type a name in the Menu title field.
7. Click the OK button to save the context menu.
8. (option) Repeat steps 2 through 5 to add more context menus.

To add a toolbar

1. In the Customize dialog click the *Toolbars* tab.
2. Do one of the following:
 - Select a menu group, then right click and choose *Append toolbar* in the context menu.
 - Select an existing toolbar in a menu group, then right click and choose *Insert toolbar* in the context menu.

New toolbars are added above the selected existing toolbar or as the first toolbar if you select the menu group.

3. Type a name in the *Title* field of the *Add toolbar* dialog.



4. Click the *OK* button.
5. (option) Adjust the Toolbar properties.

☐ Toolbar	
Title	My Drawing Tools
Alias	TB_MyDrawingTools
Position	Floating
Visible	Hide
Rows	1
Xval	
Yval	

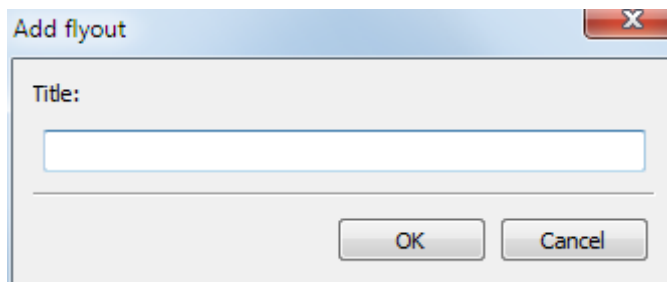
- *Title*: The toolbar name must include alphanumeric characters with no punctuation other than a dash (&endash;) or an underscore (_). The *Title* along with the *Alias* enables the toolbar to be referenced programmatically.
 - *Position*: Defines the display position of the toolbar when the CUI file is loaded. The options are: *Floating*, *Top*, *Left*, *Bottom* or *Right*. The latter four refer to the window edge where the toolbar will be docked.
 - *Display*: Defines whether the toolbar displays or not when the CUI file is loaded.
 - *Rows*: Specifies the number of rows when the toolbar position is floating. View bars with controls cannot have multiple rows.
 - *Xval*: When the position is *Floating*, specifies the X coordinate in pixels, measured from the left edge of the screen to the left side of the toolbar.
 - *Yval*: When the position is *Floating*, specifies the Y coordinate in pixels, measured from the top edge of the screen to the top of the toolbar.
6. Click the *Apply* button to save the definition(s).

NOTES The new toolbar is loaded only if the *Display* property is set to *Show*. See Opening a Toolbar for more information on opening toolbars.

To add a flyout to a toolbar

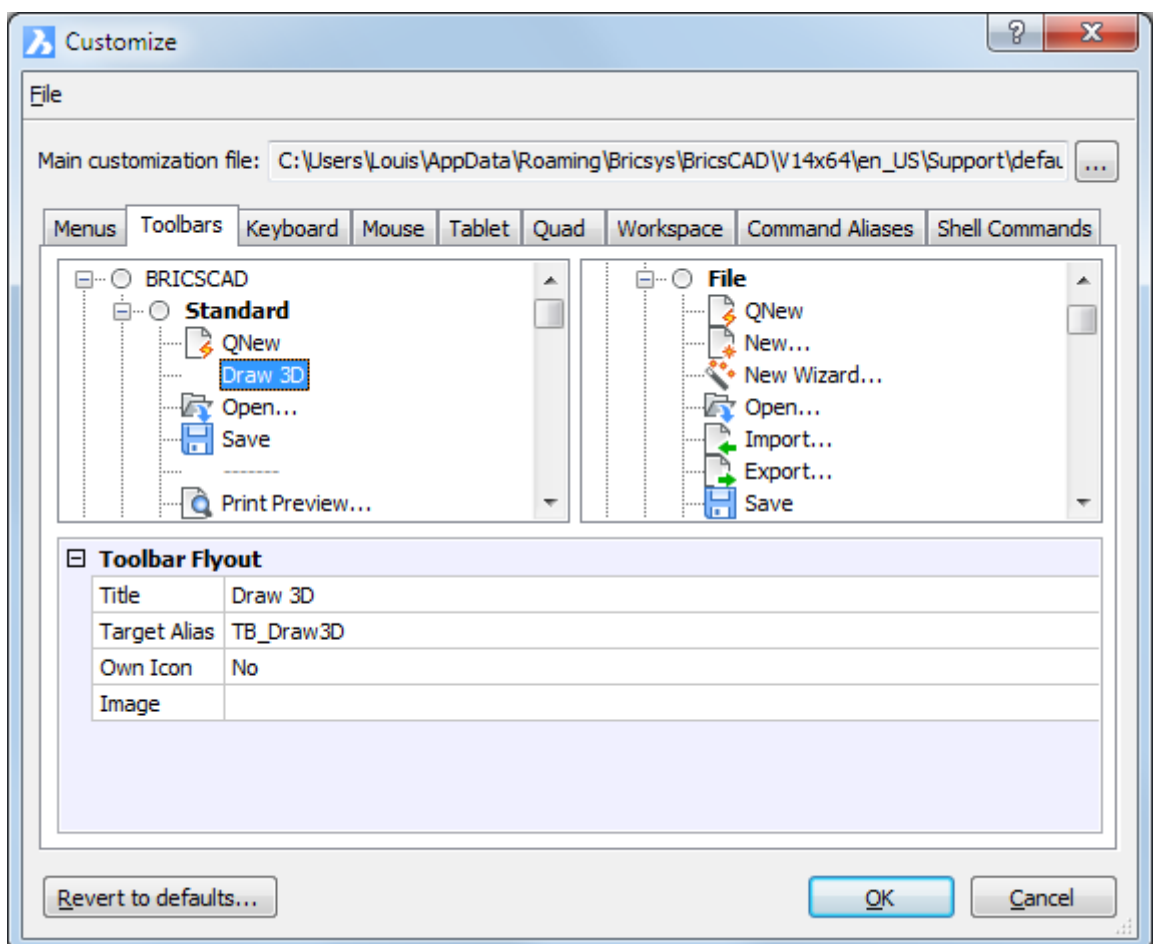
1. In the Customize dialog click the *Toolbars* tab.
2. If collapsed, expand the menu group (e.g. BRISCAD).
3. Do one of the following
 - Select a toolbar, then right click and choose *Append Flyout* in the context menu, to add the flyout at the end of the toolbar.
 - Expand the toolbar, right click a tool then choose *Insert flyout* in the context menu, to add the flyout above the selected tool.

The *Add flyout* dialog box displays:



4. Type a name in the *Title* field of the *Add flyout* dialog, then click the *OK* button.

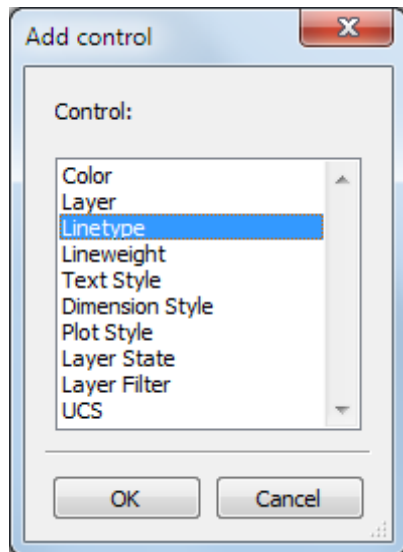
A new empty toolbar is created and inserted as a flyout in the selected toolbar:



5. Do one of the following:
 - Click the *Target Alias* field, then click the settings field and select an existing toolbar in the drop-down list.
 - Add tools to the new toolbar.

To add a control to a toolbar

1. In the Customize dialog click the *Toolbars* tab.
2. If collapsed, expand the menu group (e.g. BRICSCAD).
3. Do one of the following:
 - Select the toolbar, then right click and choose *Append control* in the context menu.
The new control will be added as the first item in the toolbar.
 - Select an existing item in the toolbar, then right click and choose *Insert control* in the context menu.
The new control will be added above the selected existing item.
4. Select a control in the *Add control* dialog.



5. Click the *OK* button.
6. To specify the width of the control, do one of the following:
 - Accept the *Default* width.
 - Type the width in pixels in the *Width* field.

☐ Toolbar Control	
Control	Dimension Style
Width	Default

NOTE Type *Default* or *0* (zero) in the *Width* field to restore the default width of the control field.

To add a keyboard shortcut

1. In the Customize dialog click the *Keyboard* tab.
2. If collapsed, expand the menu group (e.g. BRICSCAD).
3. Do one of the following:
 - Select the menu group, then right click and choose *Append shortcut* in the context menu.
The new shortcut will be added at the end of the shortcut list.
 - Select an existing shortcut in the toolbar, then right click and choose *Insert shortcut* in the context menu.
The new shortcut will be added above the selected existing shortcut.

4. Do one of the following:
 - Proceed with the Add an existing tool procedure.
 - Proceed with the Add a new tool procedure.
5. In the *Properties* grid of the new shortcut select the *Key* field, then press the key combination you want to assign to the shortcut.

Keyboard Shortcut	
Key	Ctrl+G
Tool ID	ID_GROUP_DIRECT
Help	Creates an unnamed group of the selected entities
Command	-group;;*;;
Image	custom_2

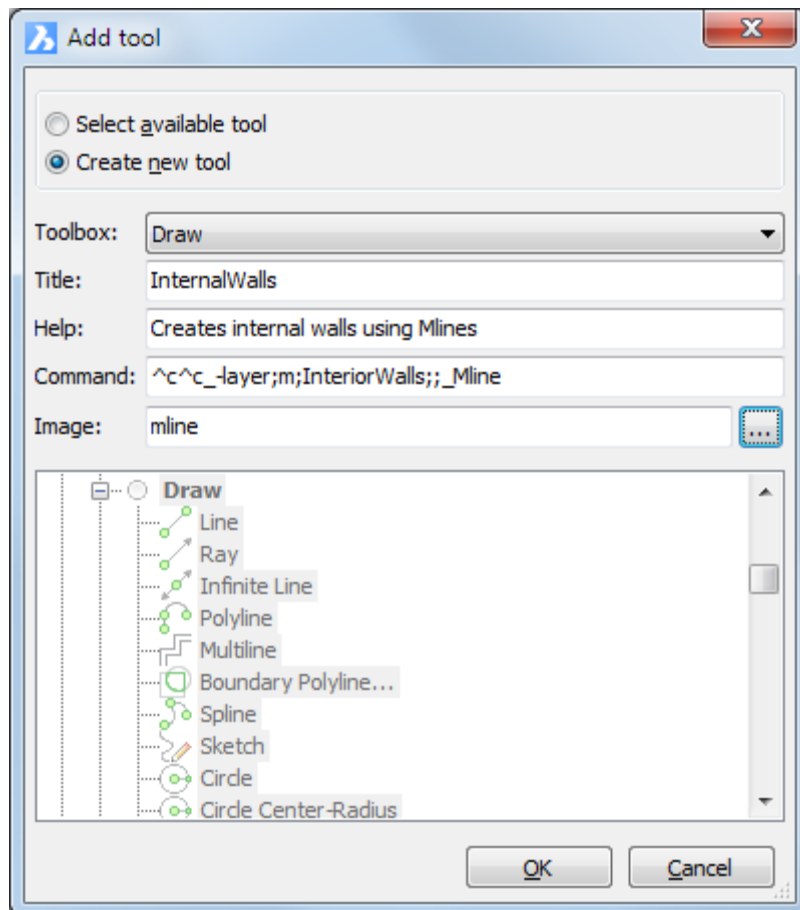
To add an existing tool to a menu, toolbar or shortcut

1. In the Customize dialog select the item you want to add an existing tool to.
2. In the *Available Tools* pane, expand the toolbox that contains the tool you want to add.
3. Drag the tool to the target item:
 - Drop the tool on top of the item name to append the tool.
 - Drop the tool on top of another tool to insert the new tool above the existing tool.
4. (option) Edit the tool in the settings grid, e.g. to add a diesel expression in the *Diesel* field.

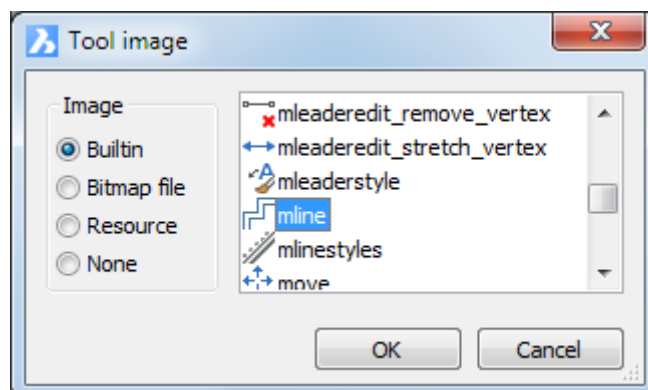
Toolbar Button	
Title	Snap to Nearest
Diesel	\$(if,\$(=,\$(and,\$(getvar,OSMODE),0x0200),0),,!.)
Tool ID	esnap_nearest
Help	Snaps to the nearest point on an entity
Command	_nearest
Image	esnap_nearest

To create a new tool in a menu, toolbar or shortcut

1. In the *Add* dialog choose *Create new tool*.
2. Select the *Toolbox* in which you want to create the new tool.



3. Type a name for the new tool in the *Title* field of the *Add tool* dialog.
4. (option) Type a help string in the *Help* field of the properties grid.
5. Type a command or command macro in the *Command* field of the properties grid. In the screenshot above the *Command* field contains a macro that makes the *InteriorWalls* layer current, then launches the *Mline* command. If the layer does not exist yet, it is created first.
6. Click the *Browse* button (...), then select an image for the new tool. Do one of the following:
 - Select *Builtin*, then choose one of the built-in icons.



- Select *Bitmap file*, and choose a bmp, gif, jpg, jpeg or png image file.
7. Click the *OK* button to create the new tool.

To change the order of the tools in a menu or toolbar

1. In the Customize dialog select the menu or toolbar.
2. Drag the tool to its new position:
 - Drop the tool on top of the item name to move it to the last position.
 - Drop the tool on top of another tool to move the tool above this tool.

Tool Palettes

Command: TOOLPALETTES

On a tool palette you can add blocks, hatches and commands in a tabbed window.

Drag a tool from the palette to the graphic screen to launch the related tool.



Tool Palettes (Internet connection needed)

To open the Tool Palettes Bar

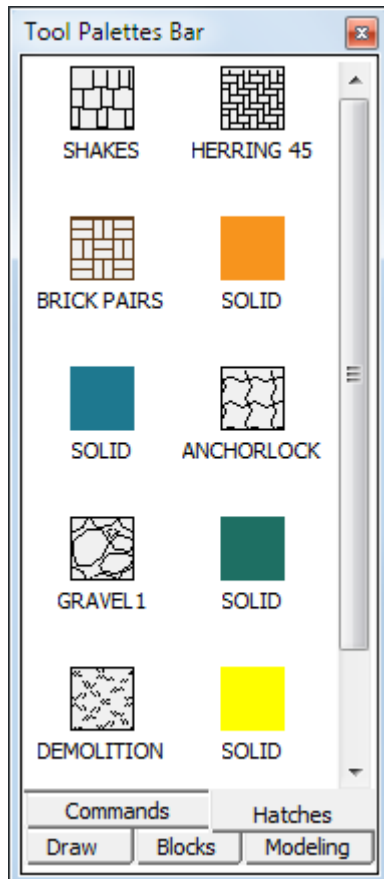
Do one of the following:

- Hover over a toolbar (Windows) or an empty toolbar area (Windows and Linux), then right click and choose *Tool Palettes Bar* in the context menu.
- Type *toolpalettes* in the command bar, then press Enter.

<input type="checkbox"/> Auto complete mode	0x000F (15)
0x0001	<input checked="" type="checkbox"/> Enable
0x0002	<input checked="" type="checkbox"/> Auto-Append
0x0004	<input checked="" type="checkbox"/> Suggestion List
0x0008	<input checked="" type="checkbox"/> Display Icons (unsupported)
0x0010	<input type="checkbox"/> Exclude the display of System Variables
Auto complete delay	0.3

To set the view options

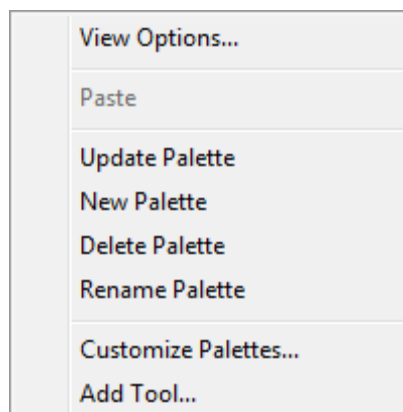
1. Right click in an empty space of the icon area on the *Toolpalettes Bar* and choose *View Options...* in the context menu.
The *View Options* dialog displays.



2. Drag the *Image Size* slider to adjust the image size.
3. Select either *Icon with text* or *Icon only*.
4. Choose either *Current Palette* or *All Palettes* from the *Apply to* list button.
5. Click the *OK* button to confirm.

To add a new tool palette

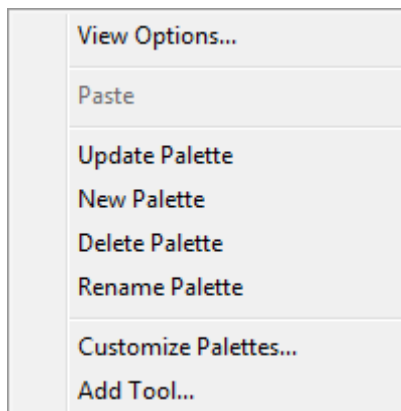
1. Move the cursor to the icon area of the *Tool Palettes Bar*, then right click. A context menu displays:



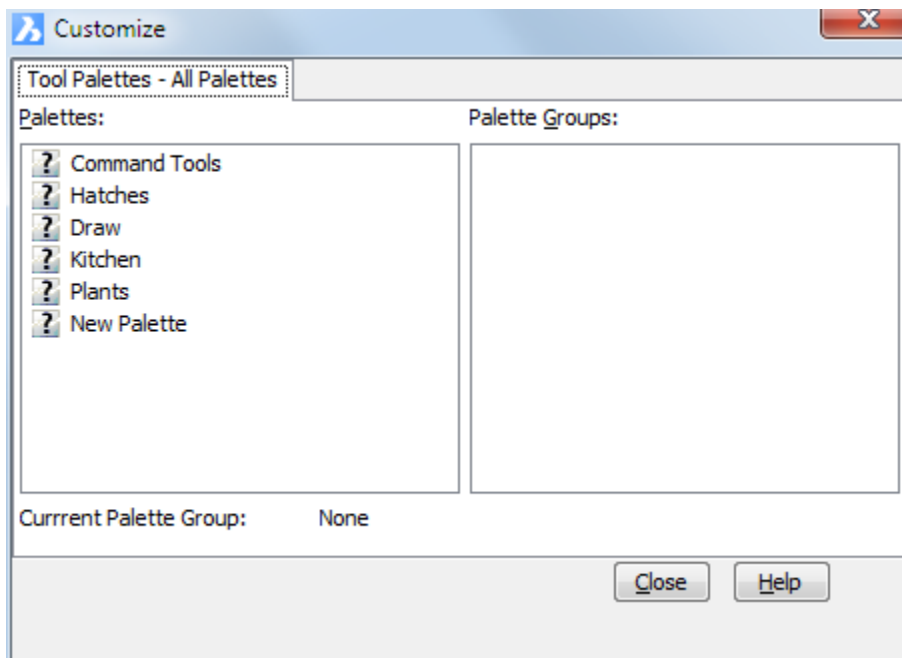
2. Choose *New Palette* in the context menu. A new tab is added.
3. Type a name for the new palette in the *Name* field.

To import a tool palette

1. Move the cursor to the icon area of the *Tool Palettes Bar*, then right click. A context menu displays:



2. Choose *Customize Palettes* in the context menu. The *Customize* dialog box displays:



3. In the *Palettes* pane on the *Customize* dialog box, right click and choose *Import* in the context menu.
4. On the *Import Palette* dialog, select a folder, then select a palette file (*.btc or *.xtp).
5. Click the *Open* button or double click to import the palette. The palette is added in the *Palettes* list.
6. Click the *Close* button to close the *Customize* dialog box.

To rename a tool palette

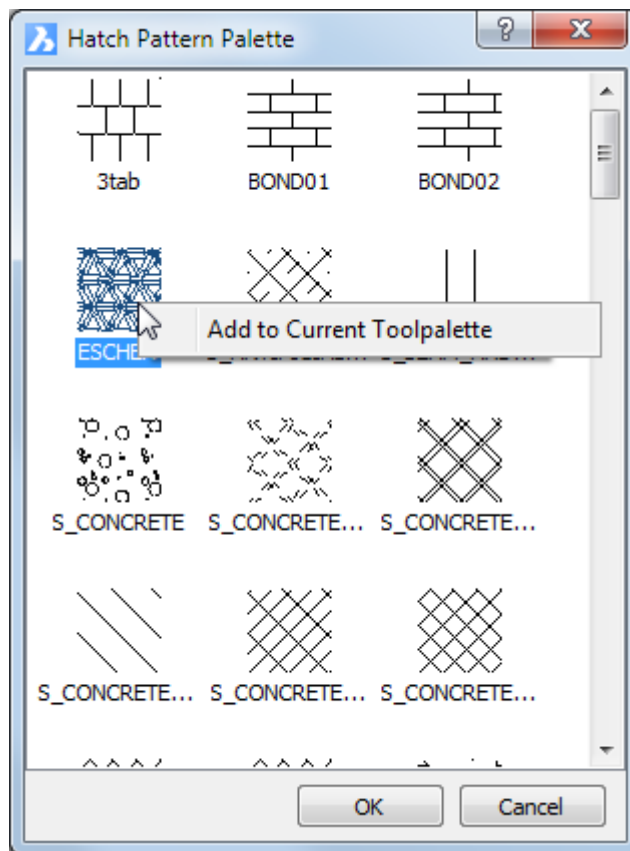
1. Right click tool palette tab and choose *Rename Palette* in the context menu.
2. Type a new name for the palette in the *Name* field.
3. Press Enter to confirm.

To delete a tool palette

1. Right click tool palette tab and choose *Delete Palette* in the context menu.
2. Click the *OK* button on the *Confirm Palette Deletion* box.

Adding a hatch pattern

1. Select the palette you want to add a hatch pattern to.
2. Open the Hatch and Gradient dialog box.
3. Select either *Predefined* or *Custom* from the *Type* list button.
4. Double click the *Swatch* field on the *Hatch and Gradient* dialog box. The *Hatch Pattern Palette* displays.
5. Select the hatch pattern you want to add, then right click.



6. Choose *Add to Current Tool Palette*. The hatch pattern is added on the current tool palette.
7. (option) Repeat steps 5 and 6 to add more hatch patterns.

Adding a block

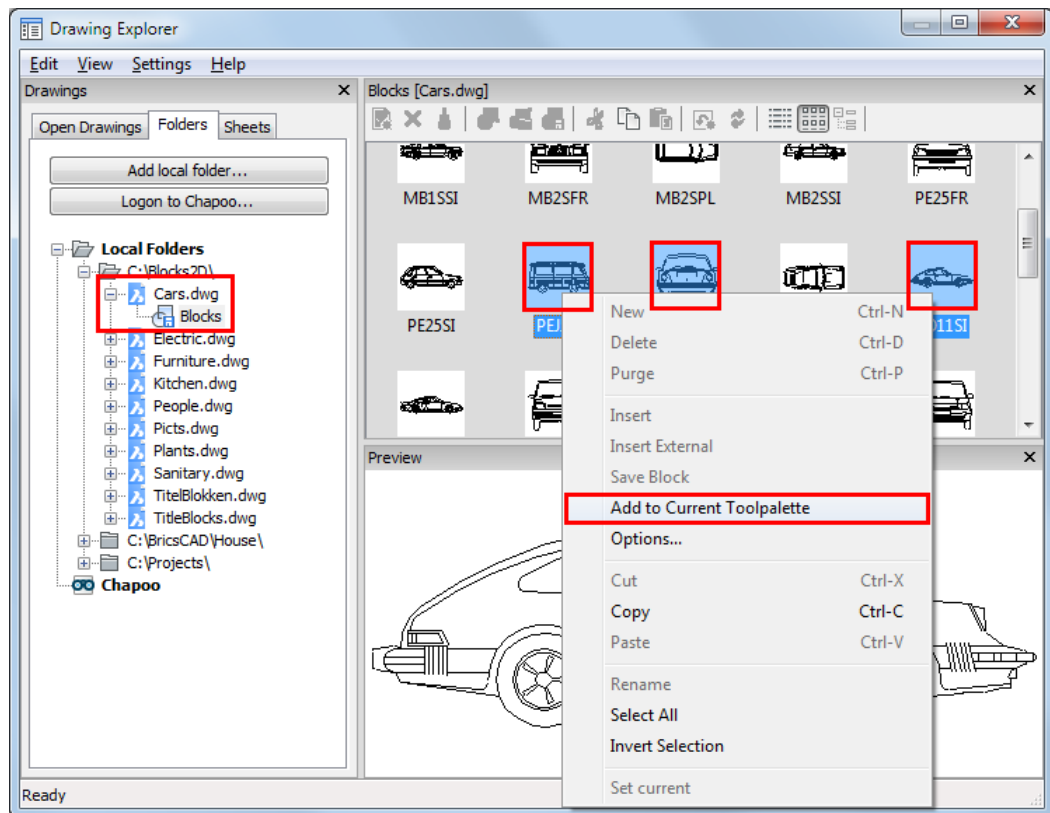
Blocks can be added from the current drawing or from any drawing.

To add a block from the current drawing:

1. Choose *Drawing Explorer - Blocks* in the *Tools* menu.
2. On the *Drawing Explorer - Blocks* dialog box select the block or blocks you want to add.
3. Right click and choose *Add to Current Tool Palette* in the context menu.

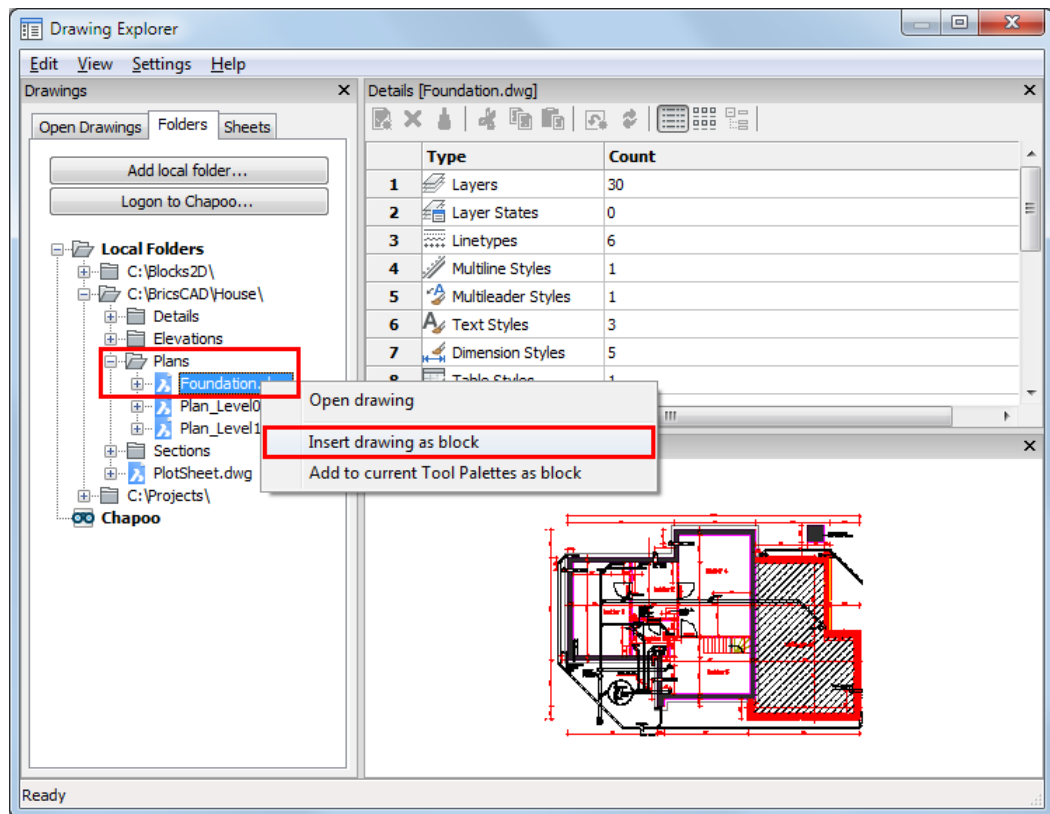
To add a block from any drawing:

1. If not already available, add the containing folder to the Folders list on the *Drawing Explorer - Drawings* dialog box.
2. Select the drawing in the containing folder on the *Drawing Explorer - Drawings* dialog box.
3. Click the *Expand* icon (+) of the containing drawing. The *Blocks* icon of the drawing displays.
4. Click the *Blocks* icon of the containing drawing. The blocks in the drawing display in the *Details* pane.
5. Select the block or blocks you want to add. Press and hold the Ctrl key to select multiple blocks.
6. Right click and choose *Add to Current Tool Palette* in the context menu.



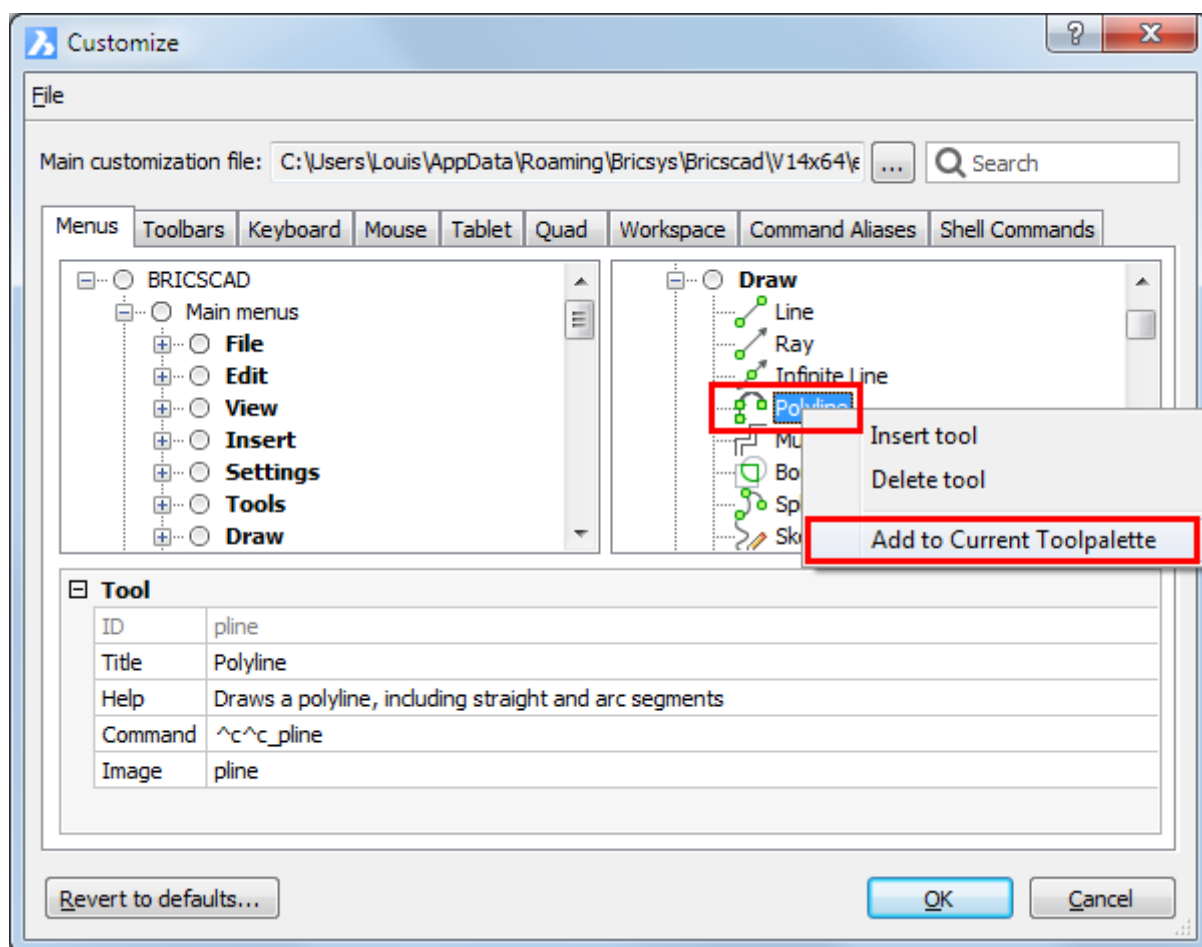
To add a drawing as a block:

1. If not already available, add the containing folder to the Folders list on the *Drawing Explorer - Drawings* dialog box.
2. Select the drawing in the containing folder on the *Drawing Explorer - Drawings* dialog box.
3. Right click and choose *Add to Current Toolpalette as block* in the context menu.



Adding a command

1. Choose *Customize* in the *Tools* menu.
2. Under *Available Tools* on the *Customize* dialog, expand the containing toolbox, then select the command in the commands list.
3. Right click and choose *Add to Current Tool Palette* in the context menu.

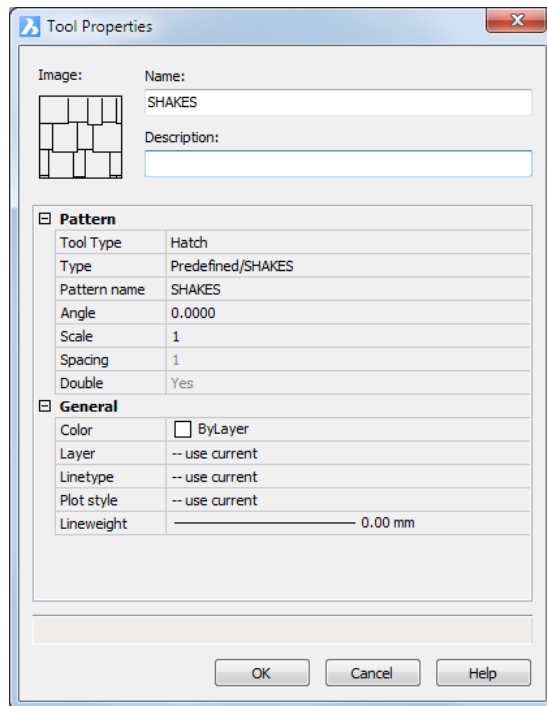


NOTE See the topic Creating Custom Tools to create your own tools, which you can then add to a tool palette.

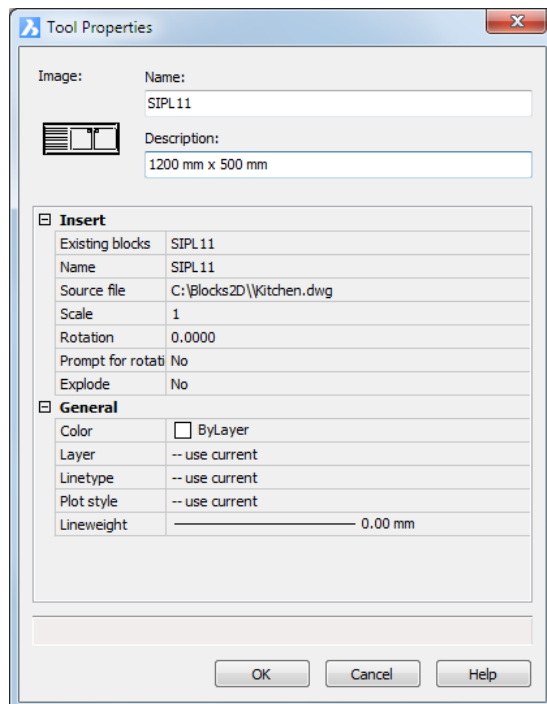
Editing a tool

General procedure to edit a tool on a tool palette:

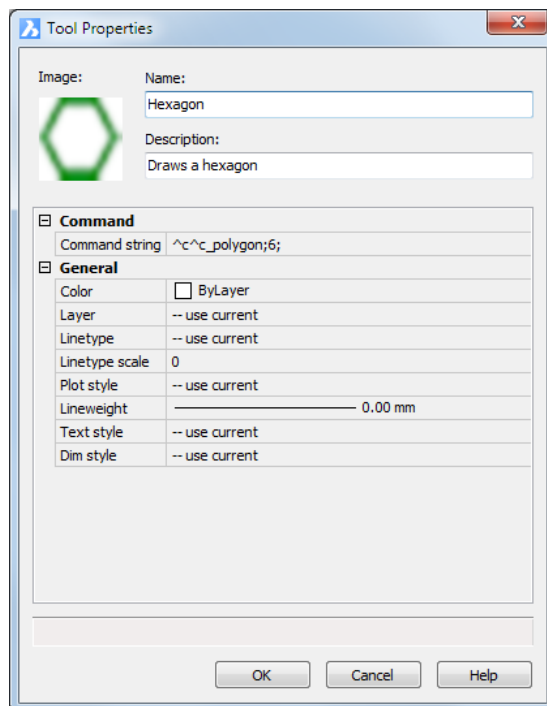
1. Right click the tool, then choose *Properties* in the context menu.
The *Tool Properties* dialog box displays.



Hatch tool properties



Block tool properties



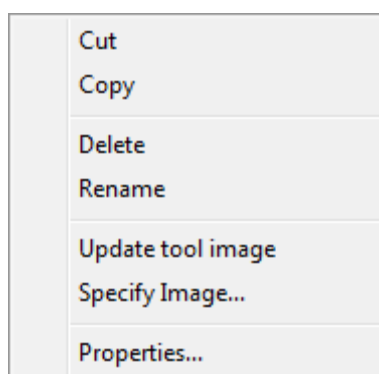
Command tool properties

2. Edit the tool properties.
3. Click the *OK* button to confirm.

Managing tools

To manage the tools on a tool palette do the following:

1. Right click the tool.
A context menu displays:



2. Choose an option in the context menu.

To specify a tool image

1. Right click the tool and choose *Specify Image* in the context menu.
A file dialog box displays.
2. Select the image file, then click the *Open* button.

NOTE To restore the original tool image: Right click the tool and choose *Remove Image* in the context menu.

To move a tool

To move a tool from one palette to another, do the following:

1. Right click the tool on the source palette and choose *Cut* in the context menu.
2. Select the target palette.
3. Right click and choose *Paste* in the context menu.
The tool is added at the bottom of the palette.

To copy a tool

To copy a tool from one palette to another, do the following:

1. Right click the tool on the source palette and choose *Copy* in the context menu.
2. Select the target palette.
3. Right click and choose *Paste* in the context menu.
The tool is added at the bottom of the palette.

Command Aliases

Command aliases are alternative names for BricsCAD commands. You can launch the LINE command by entering L at the command prompt. There is no need to type the complete name. After typing the command alias at the command prompt the command is executed. Some commands have more than one alias: e.g. REC, RECT and RECTANGLE are all aliases for RECTANG, whereas the last alias is even longer than the command name. In general, command aliases can increase your speed in entering the commands at keyboard.

Command aliases are saved in a *.pgp file (**Pro**gram **P**arameters file). By default the *default.pgp* file in the *Support* folder of the Roamable Root folder is used.

If a command can be executed transparently (= while another command is being executed), the alias can also be used transparently. To execute a command or an alias transparently, precede the command name or alias with an apostrophe (').

Command aliases cannot be used in scripts (*.scr).

To edit the alias file in a text editor

In order to modify existing aliases or to create new ones, the *default.pgp* file can be edited using an ASCII text editor (e.g. Notepad).

The first section of the *default.pgp* file defines the *Shell* commands. The second section of the *default.pgp* file defines command aliases.

The following syntax is used to define a command alias:

alias, **command*

alias = the character string to be entered at the command prompt. The command alias must be followed by a comma (,).

command = the BricsCAD command. The command name must be preceded by an asterisk (*).

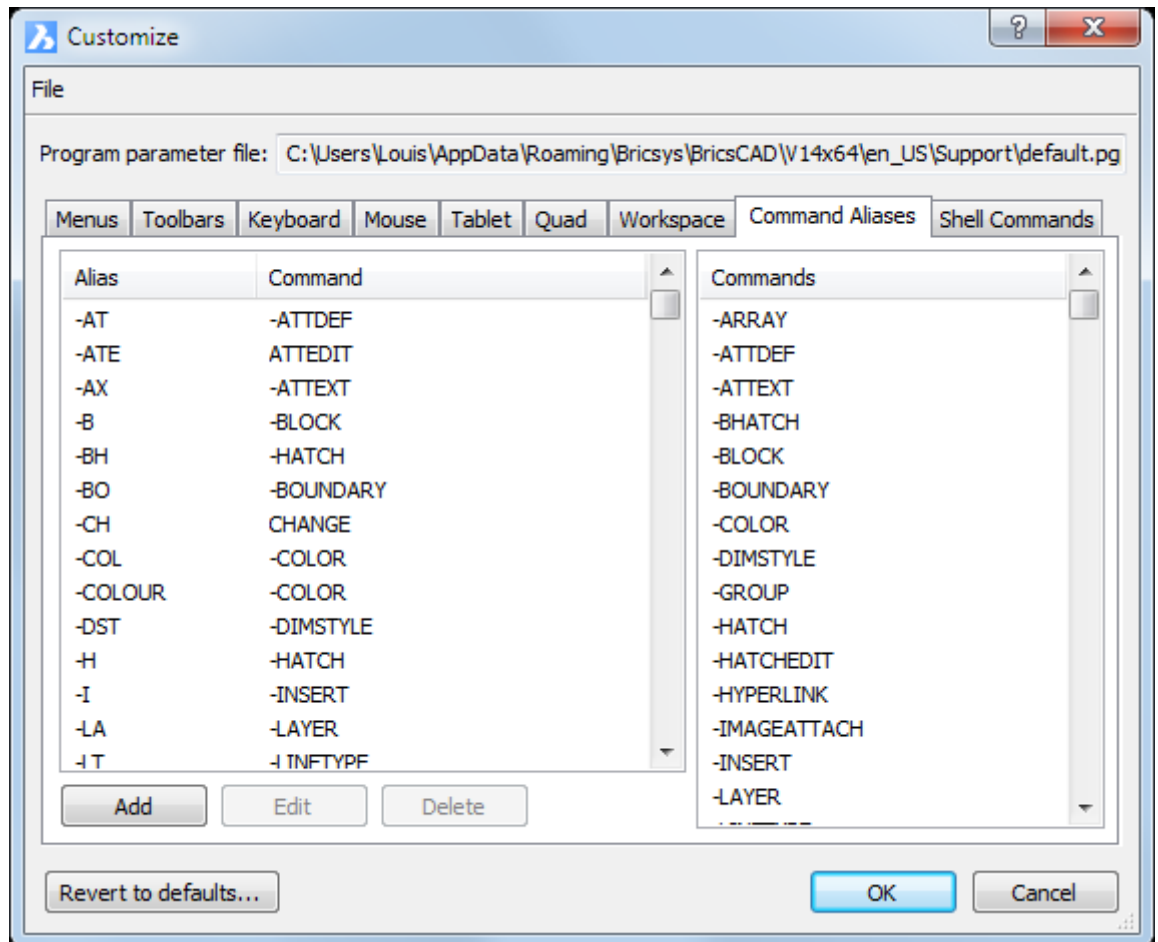
Comment lines must be preceded by a semicolon (;). Comment lines can be used to add additional information, e.g. when the file was last edited and by whom.

Some commands, such as HATCH or BOUNDARY, can be preceded by a hyphen (-) to access the command line version of the command instead of the dialog box version. Command aliases can be created for these commands also, e.g.: *vi*, **-view*

Execute the REINIT command to reload the alias file when it was externally edited.

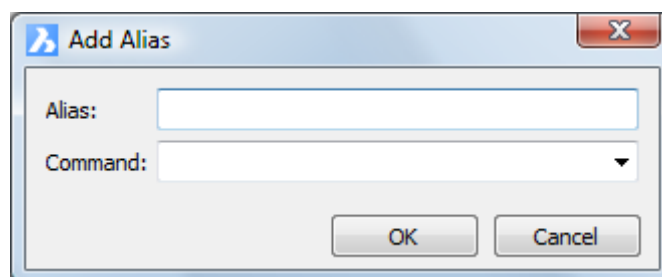
Using the Customize dialog to edit the alias file

1. Open the Customize dialog.
2. Click the *Aliases* tab on the *Customize* dialog.



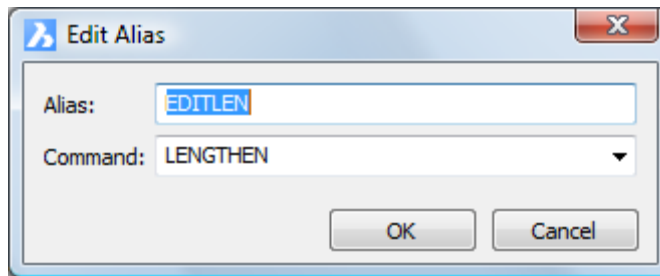
3. (option) To create a new alias:
 - Click the *Add* button.

The *Add Alias* dialog box displays:



- Click the down arrow at the right hand side of the *Command* field, then select a command.
You can start typing the command name in the *Command* field to easily find the command in the list.
- Type the command alias for the command in the *Alias* field.
- Click the *OK* button.

4. (option) To edit an existing alias:
 - Select the alias in the list.
 - Click the *Edit* button.
 The *Edit Alias* dialog box displays:



- Type a new alias in the Alias field.
If the alias already exists you are prompted to overwrite the existing alias.
 - Click the *OK* button.
5. (option) To delete an alias:
 - Select the alias in the list.
 - Click the *Delete* button.
 A *Confirm* box displays.
 - Click the *OK* button on the Confirm box to delete the alias.
 6. Click the *OK* button on the *Customize* dialog to stop.

NOTE If you use the *Customize* dialog to edit the alias file, it is not necessary to execute the REINIT command.

User Profile Manager

Command: PROFILEMANAGER

In BricsCAD you can customize the settings that control your drawing environment, which can then be saved in a *user profile*. You can create multiple user profiles, e.g. a profile to run BricsCAD with a white background and a limited set of toolbars and another profile with no command bar and all toolbars and snap settings needed to draw comfortably in 3D. User profiles can also be used when multiple users share the same computer and each user likes to create his personal drawing environment.

User profiles can be saved to a file, which can then be imported on a different computer. This allows you to copy your settings to a new computer or to install the same settings on multiple computers.

If you use a third party applications on top of BricsCAD, you might want to create a user profile to use with each of these applications.

The *User Profile Manager* is a standalone tool that lists the existing profiles and provides all the tools needed to Load, Create, Set Current, Copy, Rename, Delete, Export and Import user profiles.

Content of a user profile

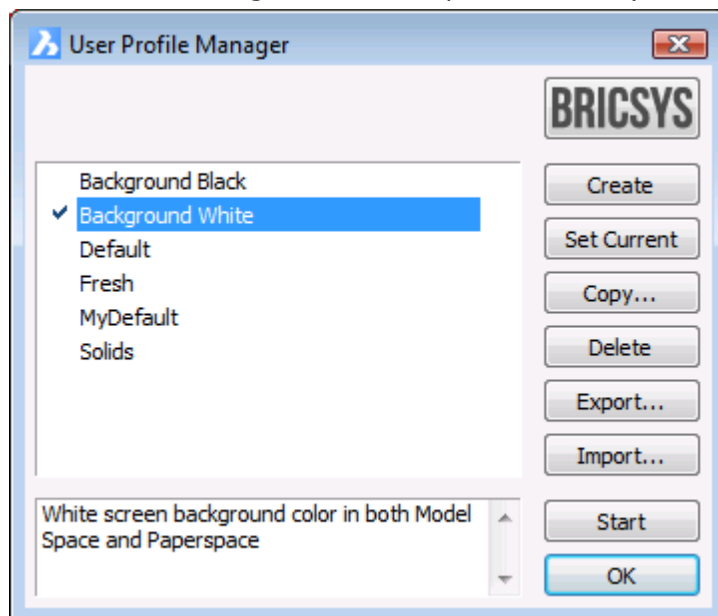
User profiles save many settings that control the drawing environment. Once you start BricsCAD using a user profile, it automatically tracks and stores changes that you make to your drawing environment.

- Configuration settings: all settings listed in the *Settings* dialog and which are saved in the registry (🔧).
- Plot settings in Model Space.
- Settings in dialog windows, e.g. search options in the *Settings* dialog.
- Project settings
- Recent paths
- Status bar settings
- 'Tip of the Day' settings
- Toolbar settings

To Launch the User Profile Manager

Do one of the following:

- When BricsCAD is running: choose *User Profile Manager* in the *Tools* menu.
- From the *Windows Start* menu, choose *All Programs > Bricsys > BricsCAD > User Profile Manager*.
- Double click *userprofilemanager.exe* in the BricsCAD program folder (e.g. C:\Program Files\Bricsys\BricsCAD Vx).



In the *User Profile Manager* dialog:

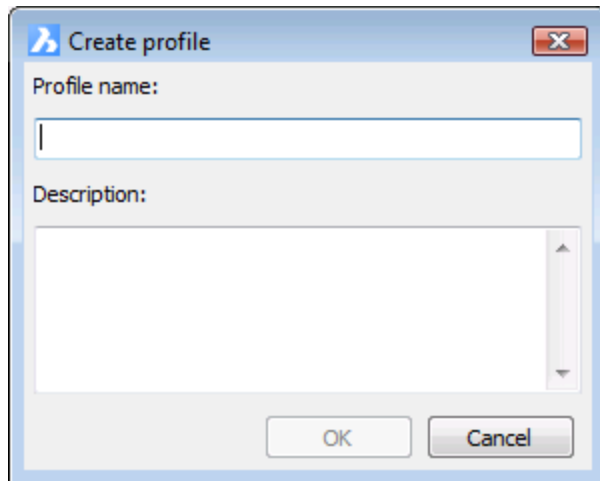
- The *Current Profile* is marked.
- *Create*: Creates a new profile, using default settings.
- *Set Current*: Sets the selected profile as the current profile, which will be used the next time you start BricsCAD.
- *Copy*: Click to make a copy of the selected profile.
- *Delete*: Click to delete the selected profile.
- *Export...*: Click to save the selected profile to a file on your computer, disk, or network. The file is saved with an *.arg* extension.
- *Import...*: Click to open a profile that has been saved to *.arg* file).
- *Start*: Starts BricsCAD with the selected user profile.

- OK: Closes the *User Profile Manager* dialog
- Double click a profile in the list of the available profiles to start BricsCAD using this profile.

Creating user profiles

To create a new user profile:

1. Launch the User Profile Manager.
2. Click the *Create* button on the *User Profile Manager* dialog window. The *Create Profile* dialog opens.



3. Type a name for the new profile in the *Profile name* field.
4. (option) Type a description for the new profile in the *Description* field.
5. Click the *OK* button to create the profile. The new profile is added in the available profiles list. The new profile is created using default hard-coded values for all settings.

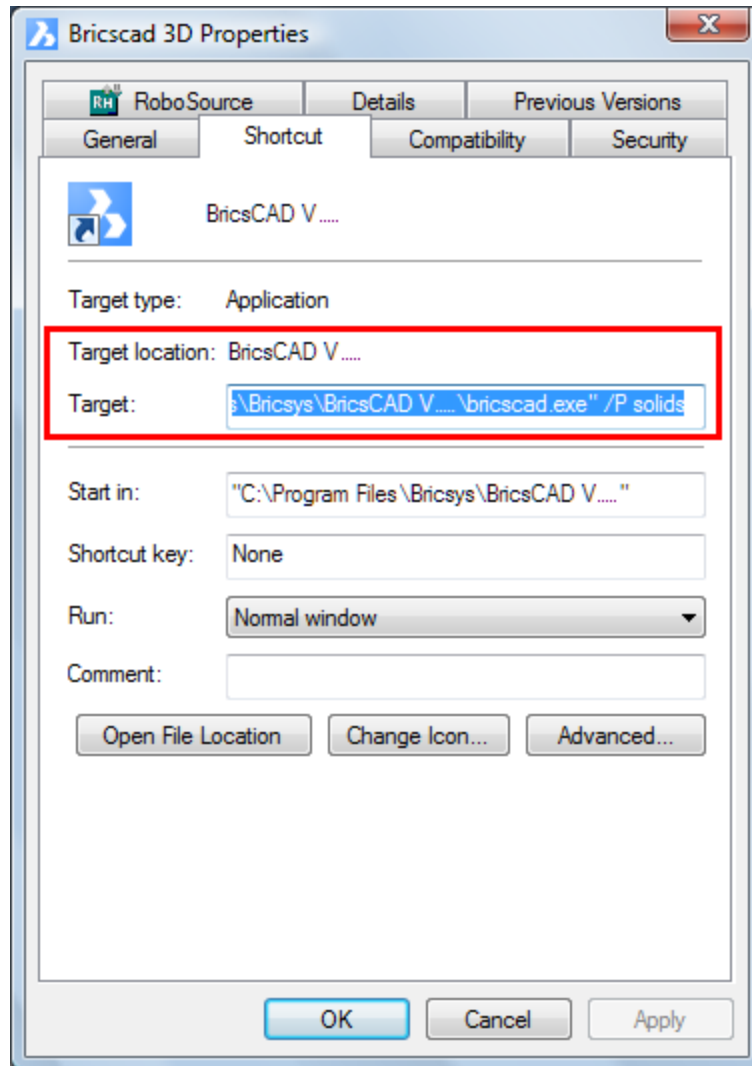
NOTE Use the Copy option to create a new user profile as a copy of an existing profile.

Starting BricsCAD with a specific user profile

When you start BricsCAD, the user profile that was current when you closed the previous BricsCAD session is used.

To launch BricsCAD using a specific user profile, do one of the following:

- Add the */P option* to the *Target* settings of the BricsCAD shortcut. To open the shortcut settings dialog: right click the shortcut icon on the desktop and choose *Properties* in the context menu. A typical BricsCAD shortcut could be `"C:\Program Files\Bricsys\BricsCAD Vx\bricscad.exe" /P <UserProfileName>`



- Launch the User Profile Manager, then either double click a user profile or select a user profile and click the *Start* button.

NOTE

- The */P* option is case insensitive.
- If you load a user profile when BricsCAD is already running a second instance of BricsCAD is launched, at least on condition that the *SINGLETON* variable is OFF. This variable controls whether multiple BricsCAD sessions can run simultaneously. See the Settings dialog for more information about variables and settings.
- If the user profile in the argument of the */P* option does not exist, the user profile is created automatically using default hard-coded values for all settings.
- If the user profile name contains spaces, use double quotes: e.g. */P "background white"*.

Restarting BricsCAD using a different user profile

1. Choose *User Profile Manager* in the *Tools* menu.
2. Select the user profile you want to start up with.
3. Click the *Set Current* button on the *User Profile Manager* dialog.
4. Close and restart BricsCAD.

Managing user profiles

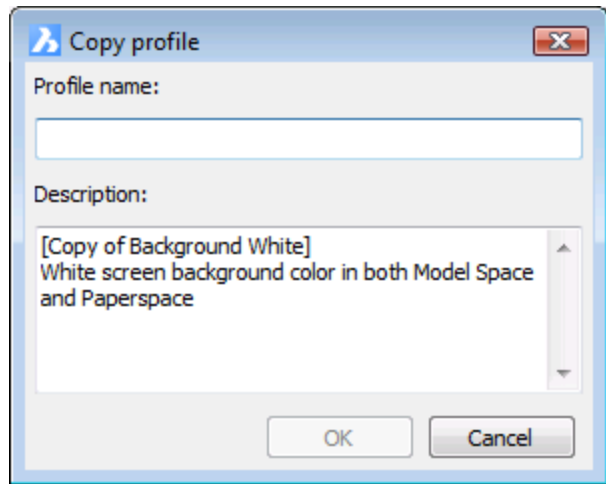
Once you start using user profiles, you may need to copy, rename or delete them. Copying a profile is a quick way to create a new profile based on an existing profile.

To *copy* a user profile:

1. Launch the User Profile Manager.
2. Select the user profile you want to copy.

Click the *Copy* button.

The *Copy Profile* dialog displays.



3. Type a name for the new profile in the *Profile name* field.
4. (option) Type a description for the new profile in the *Description* field.
5. Click the *OK* button to create the profile.

To *rename* a user profile:

1. Launch the User Profile Manager.
2. Select the user profile you want to rename.
3. Click the name of the selected user profile.
The *Profile name* field is active now.



4. Type a new name in the *Profile name* field.
5. Click outside the *Profile name* field.

To edit the *Description* field:

1. Select the profile you want to edit the description of.
2. Edit the *Description* field.

To *delete* a user profile:

1. Launch the User Profile Manager.
2. Select the user profile you want to delete.
3. Click the *Delete* button.
4. Click the *Yes* button on the confirmation box to delete the profile.

NOTE The *current profile* cannot be deleted.

To export a user profile

1. Launch the User Profile Manager.
2. Select the profile you want to export in the profile list.
3. Click the *Export* button.
The *Save As* dialog displays.
4. Select a folder to save the user profile in.
5. (option) Edit the name of the profile in the *File name* field.
6. Click the *Save* button.
The file is saved with an **.arg* extension.

To import a user profile

1. Launch the User Profile Manager.
2. Click the *Import* button.
The *Open* dialog displays.
3. Select a user profile file (*.arg).
4. Click the *Open* button.
The *Import Profile* dialog displays.
5. (option) Type a different name in the *Profile name* field.
6. (option) Edit the profile description.
7. Click the OK button on the *Import Profile* dialog.
The profile is added to the profile list.

To import a user profile from another user

When you import a user profile of another user all user dependent search paths (e. g. the *local root* and *roamable root* folders - see BricsCAD User Files) still refer to the other user, as a result the folders cannot be found and error messages such as "Unable to open program parameters file: default.pgp" might pop up when you start BricsCAD.

To correct this problem do the following:


1. Open the exported user profile (*.arg) in a text editor such as Notepad.
2. Replace the user name dependent references with an environment variable.
On a Windows XP computer replace *C:\\Documents and Settings\\username* with *%USERPROFILE%*.
On a Windows Vista or Windows 7 computer replace *C:\\Users\\username* with *%USERPROFILE%*.
Use the *Find/Replace* procedure to quickly replace all references.
3. Save the user profile (*.arg)
4. Import the user profile.
5. Start BricsCAD using the imported profile.
The environment variable *%USERPROFILE%* will be replaced with:
 - *C:\\Documents and Settings\\currentusername* on a Windows XP computer.
 - *C:\\Users\\currentusername* on a Windows Vista or Windows 7 computer.

NOTE Environment variables in profiles, using the %<variable_name>% format, are supported in V10.3.11. and higher.

Projects

Project names are especially useful to manage xrefs and images when exchanging drawings. If external references (xrefs) and images are not found in the saved path, the project search paths are used to find the external references and images. Project names are saved in the registry and contain one or more search paths.

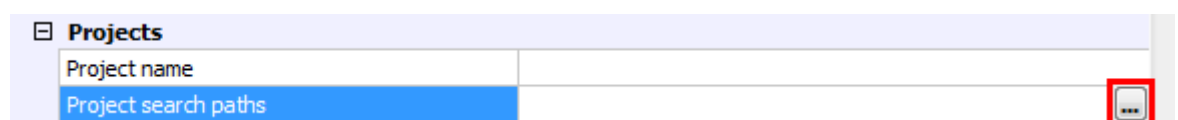
Opening the Project settings

1. Click the *Settings* tool button () on the *Standard* toolbar.
or
Choose *Settings...* in the *Settings* menu.
The *Settings* dialog opens.
2. Go to *Program Options > Files > Projects > Project search paths*.

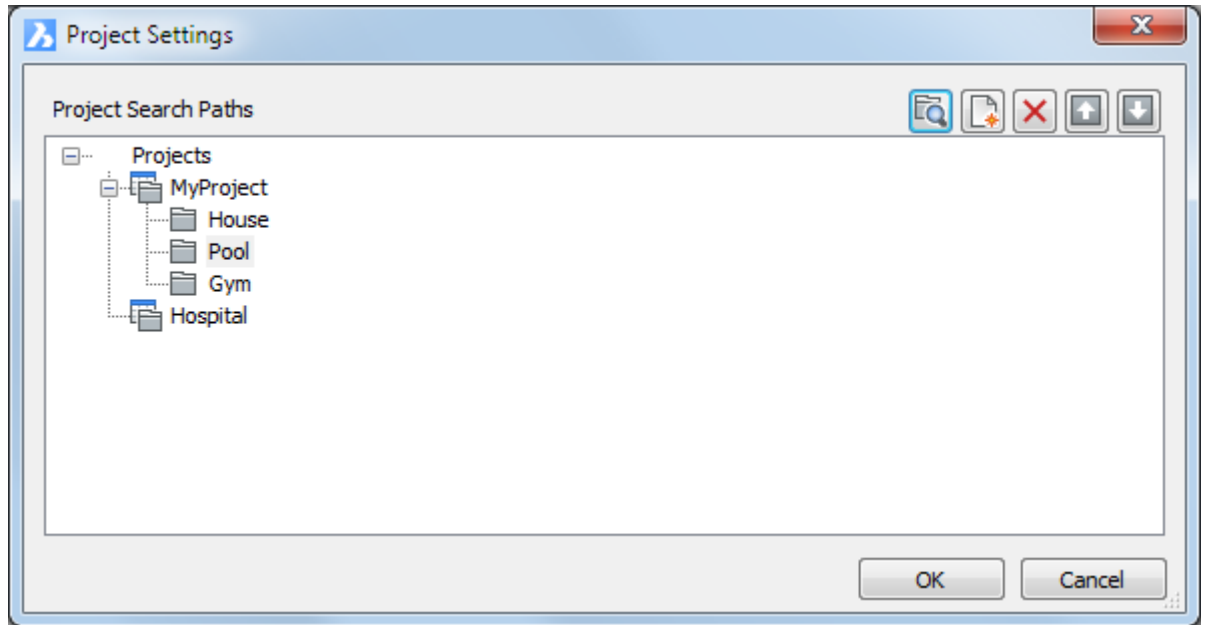
Creating projects

To create a project:

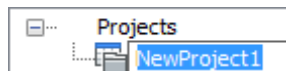
1. Open the Project settings.
2. Click the *Browse* button at the right hand side of the *Project search paths* settings field.



The *Project Settings* dialog opens.



3. To create the first project:
 - Click the *New* button (📁+) on the *Project Settings* dialog.

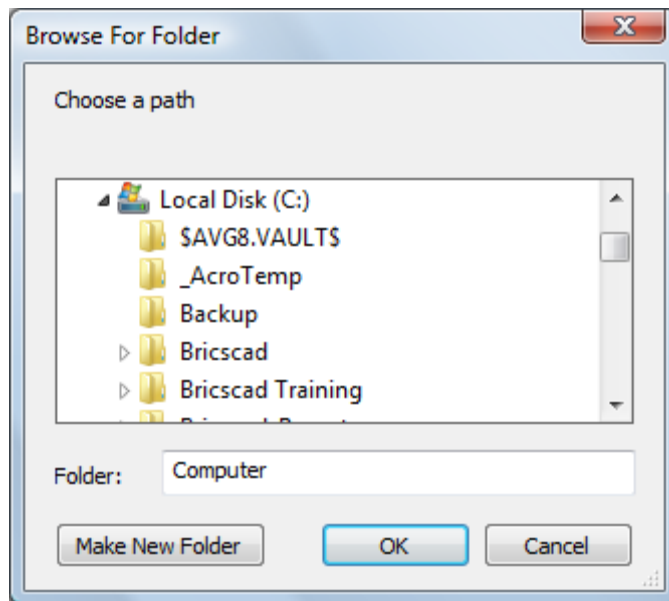


- Type a name to replace the *NewProject1* default name.
4. To create additional projects.
 - Click *Projects* (on top of the projects tree) then click the *New* button (📁+).
 - Type a name to replace the *NewProject1* default name.

Adding search paths to a project

To add a search path to a project:

1. Open the Project settings.
2. Select the project in the *Projects* tree.
3. Click the *Browse* button (🔍).
The *Browse for folder* dialog opens.



4. Do one of the following:
 - Select a folder.
 - Select a parent folder, then click the *Make New Folder* button to create a new folder.
5. Click the *OK* button to add the selected folder to the project search paths.

Changing the order of the search paths

1. Open the Project settings.
2. Click the *expand* button (+) of the project you want to edit.
3. Select a search path, then click the up (↑) or down (↓) buttons to change the order of the search paths.

Assigning a project to the current drawing

1. Open the Project settings.
2. Type the name of an existing project in the *Project name* settings field (*PROJECTNAME* system variable).

Drawing Accurately



Drawing Accurately (Internet connection needed)

In BricsCAD the following drawing aids are available:

- Dynamic dimensions
- Coordinate Input
- Snap and Grid
- Drawing Limits
- Ortho Mode
- Entity Snaps
- Polar Tracking
- Entity Snap Tracking
- User Coordinate Systems
- Direct Distance Entry

Using Orthogonal Mode

Orthogonal Mode restricts the movement of the cursor to be parallel to the X-axis or the Y-axis of the current coordinate system. For example, with the default 0-degree orientation (angle 0 at the "three o'clock" or "east" position), when the *Orthogonal Mode* setting is enabled, lines are restricted to 0 degrees, 90 degrees, 180 degrees, or 270 degrees. As you draw lines, the rubber-banding line follows either the horizontal or vertical axis, depending on which axis is nearest to the cursor: type the length of the line in the command bar, then press Enter.

NOTES

- When you enable the Isometric snap style, cursor movement is restricted to orthogonal equivalents within the current isometric plane.
- You can rotate the *Orthogonal Mode* axes using the Snap Angle setting.
- Press and hold the Shift key to reverse the *Orthogonal Mode* setting.
- *Orthogonal Mode* is overruled by entity snaps.
- Turning on *Ortho Mode* automatically disables Polar Tracking and vice versa.

To toggle Orthogonal Mode

Do one of the following:

- Click the *ORTHO* field in the Status Bar.
- Press the *F8* function key.
- Press and hold the Shift key.
- Type *ortho* in the command bar, then choose the appropriate setting.
- Set the *Orthogonal Mode* in the *Settings* dialog.

Direct Distance Entry


Direct Distance Entry (DDE) means that you can type the length of lines, polyline segments, circle radii and diameters, movement vectors in commands such as Move, Copy and Stretch, etc. in the command bar.

When combined with either Orthogonal Mode or Polar Tracking, *Direct Distance Entry* is a particularly efficient method to draw lines of a specified length and direction, and to move or copy objects over a specified distance.

To help you keep track of the position of the cursor it is recommended to set the readout of the coordinate field in the Status Bar to *Relative*.

To define the Coordinates setting

Do one of the following:

- Right click the  then choose *Relative* in the context menu.
- Click the *Coordinates* field in the status bar to cycle the *Coordinate* setting. The sequence is *Off*, *Relative*, *Absolute*.
- Press the *F6* function key or use the *Ctrl + I* keyboard shortcut to cycle the *Coordinate* setting. The sequence is *Off*, *Relative*, *Absolute*.
- Type *coords* in the command bar, then press Enter.
Type 2 and press Enter.
- Open the Settings dialog and expand the *Display/Viewing* settings sub-category under *Drafting* in the *Drawing* settings class.
Set the *Coordinates* setting to *Coordinates in polar form for point, distance and angle selection*.



Unit Settings

Commands: UNITS and -UNITS

The *Units* command sets units of linear and angular measurements through the Settings dialog box.

The *-Units* command sets units of linear and angular measurements in the command bar.

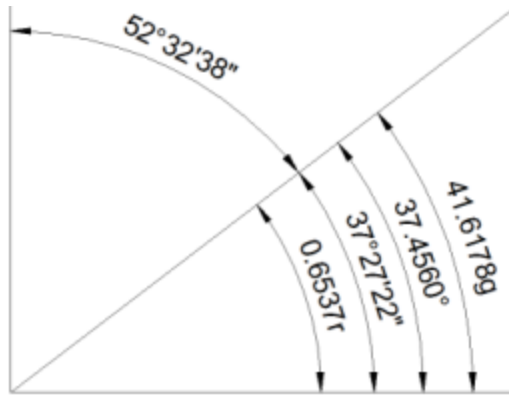
The INSUNITS system variable defines a drawing units value for automatic scaling when inserting blocks or attaching Xrefs.

Entering angles

The *AUNITS* system variable defines the angular unit type.

You can enter angles in any of 5 formats:

- **Decimal degrees:** 37.456 (AUNITS = 0)
- **Degrees / Minutes / Seconds:** 37°27'22" or 37d27'22" (AUNITS = 1)
- **Gradians:** 41.6178g (AUNITS = 2)
- **Radians:** 0.6537r (AUNITS = 3)
- **Surveyor's Units:** N52°32'38"E or N52d32'38"E (AUNITS = 4)
Surveyor's units are measured from the North (N) or the South (S) to the East (E) or to the West (W).



The Quad cursor menu

The Quad cursor menu, aka the Quad is an alternative to grip-editing of entities, offering a (much) richer set of editing operations while requiring fewer clicks, without cluttering the screen with loads of grip-glyphs.

While the Quad was initially conceived for easy and fast editing of 3D solids, in V14 the Quad is extended to allow enhanced editing of all entity types.

To activate the Quad

Activation of the Quad is controlled by the QUADDISPLAY system variable.

To toggle the Quad On/Off, do one of the following:

- Click the *QUAD* field in the Status bar.
- Press the *F12* function key.
- Edit the *QUADDISPLAY* system variable in the Settings dialog.
- Type *quaddisplay* in the command bar, then choose an option.

Editing the Quad settings

1. Right click the *QUAD* field in the Status bar, then choose *Settings* in the context menu.

The *Settings* dialog opens, showing the *Quad settings*:

Quad Settings	
Quad aperture size	20
Quad default command launch	[1] Launch default quad command on mouse right-click when quad is displayed
Quad command sorting	[0] Do not sort
Quad	<input checked="" type="checkbox"/> Display quad
Quad width	4
Quad icon size	[16] Small icons
Quad icon space	[1] Normal
Quad show delay	100
Quad hide margin	30
Quad hide delay	1000
Quad tooltips	<input checked="" type="checkbox"/> Display quad tooltips
Quad transparency delay	2000
Quad transparency factor	75
Quad expand group	[0] Expand on mouse enter
Quad warp pointer	[1] Warp pointer so it stays on same position relative to group button

- *Quad aperture size*: Distance in pixels to search for neighboring entities.
- *Quad default command launch*: Sets the action to launch the default Quad command.
- *Quad command sorting*:
 - *Sort commands within groups*: most recently used command first.
 - *Sort command groups*: most recently used group on top.
- *Quad*: Controls the Quad display.
- *Quad Width*: Defines the number of commands in a row. Default = 4 commands.
- *Quad icon size*: Small icons (16 x 16 pixels) or Large icons (32 x 32 pixels)



Small Icons



Large Icons

- *Quad icon space*: Defines the size of the Quad tool buttons: small, normal or large.



Narrow



Normal



Wide

- *Quad show delay*: Sets the time lap in milliseconds between the highlighting of the entity and the Quad display.
- *Quad hide margin*: Sets the distance in pixels the cursor can move away from the Quad. If the cursor moves outside this limit, the Quad is hidden.
- *Quad hide delay*: Sets the time lap in milliseconds to hide the Quad when the cursor is outside the *Quad hide margin* limit.
- *Quad tooltips*: Controls the display of command tooltips.
- *Quad transparency delay*: Sets the time lap in milliseconds before the Quad display becomes transparent.
- *Quad transparency factor*: Sets the Quad transparency factor in percent: 0 = Opaque, 100 = fully transparent.

- *Quad expand group*: Defines when a Quad command group is expanded: on mouse enter or on mouse click.
 - *Quad warp pointer*: Defines what to do to keep the position of the pointer and the cursor in sync when a command group expands or collapses: move the Quad or move the pointer.
2. To edit a setting, select the setting then do one of the following:
 - Type a new value.
 - Click the check box.
 - Choose an option.
 3. Close the *Settings* dialog.

The Quad layout

The layout of the Quad cursor menu depends on:

- The entity type under the cursor.
- The current Workspace.
- Geometric events such as intersection or tangency.

In general the Quad has three appearances: last command, command history and full.

- When the cursor hovers over an entity the Quad displays the command most recently used with this entity type (1).



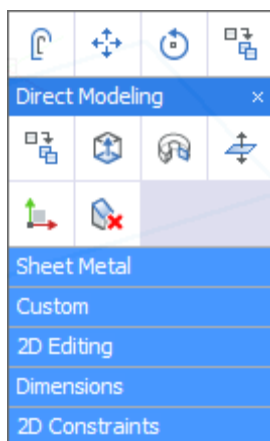
Right click to launch the command.

- Move the cursor to the Quad, to display the four most recently used commands with this entity type.



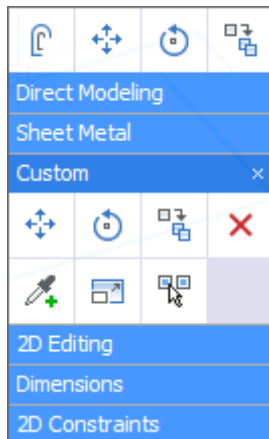
Click an icon to launch the command.


- Move the cursor over the blue field below the icons to expand the Quad cursor menu.
The first command group expands.

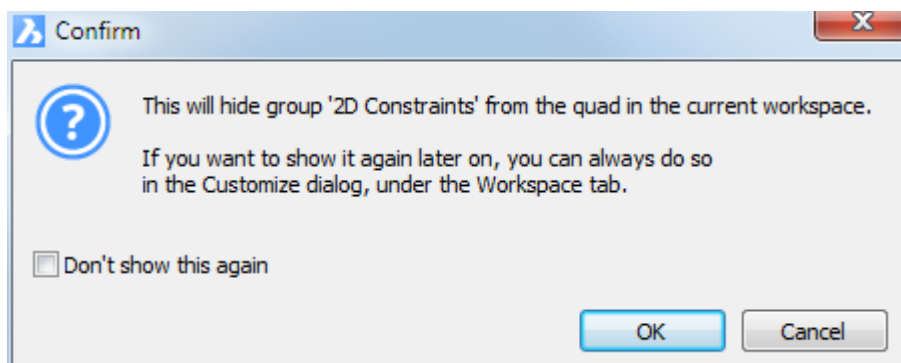


Click an icon to launch a command.

Depending on the current value of the *QuadExpandGroup* user preference either click another command group or move the cursor to another command group.



- (option) Click the *Close* icon () of a command group to hide this command group in the Quad for the current workspace. Click the *OK* button on the *Confirm* dialog box to hide the command group.



See To edit the workspace command groups to switch closed command groups on.

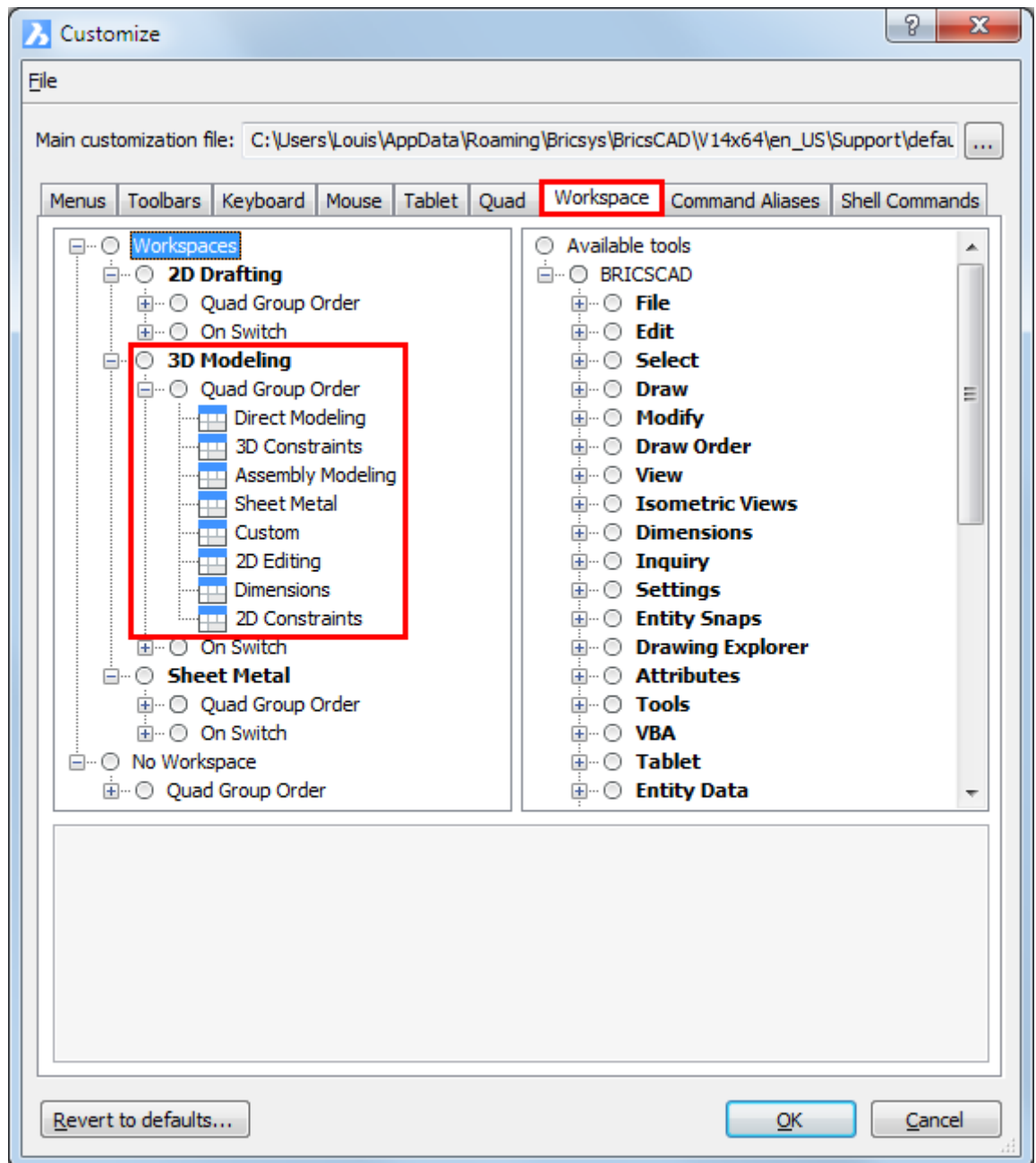
Customizing the Quad

The number of available command groups in the Quad is defined through the current workspace.

Only the content of the *Custom* command group can be modified. The content of the other command groups is hard-coded and cannot be modified.

To edit the workspace command groups

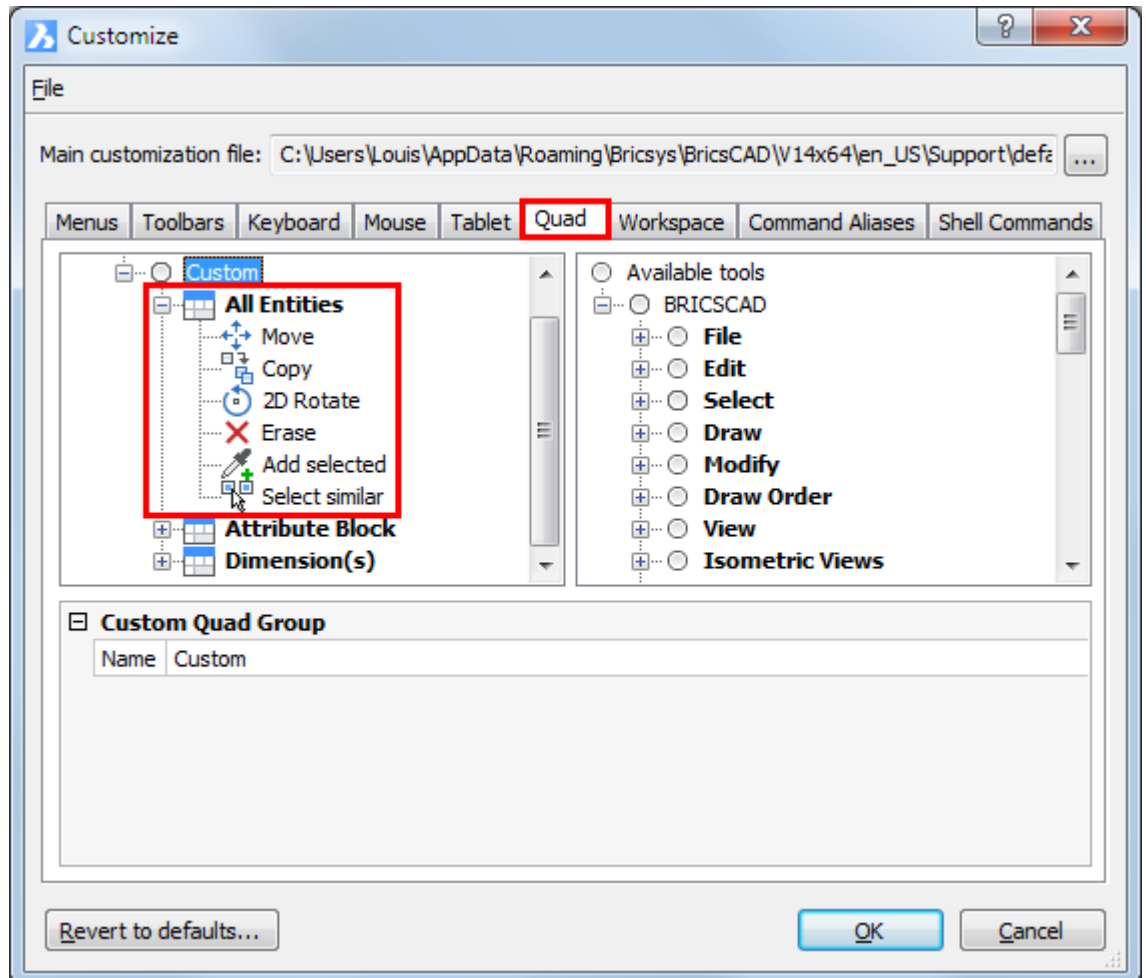
1. Move the cursor over a toolbar, then right click and choose *Customize* in the context menu.
2. Click the *Workspace* tab on the *Customize* dialog box.
3. Expand the workspace you want to edit.
4. Expand the *Quad Group Order* list.
Command groups that are currently switched off display in grey.



5. To edit visibility state of a command group, right click a command group then choose *Switch On* or *Switch Off* in the context menu.

To edit the Custom command group

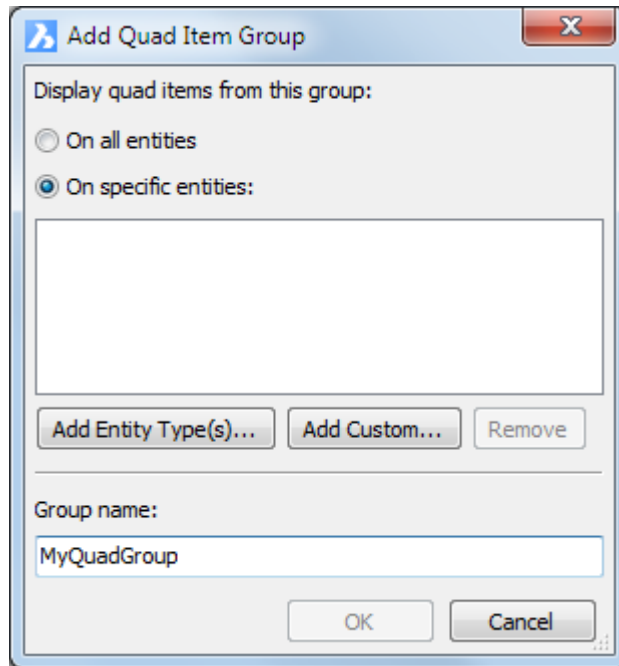
1. Move the cursor over a toolbar, then right click and choose *Customize* in the context menu.
2. Click the *Quad* tab on the *Customize* dialog box.
3. Expand the Custom command group.



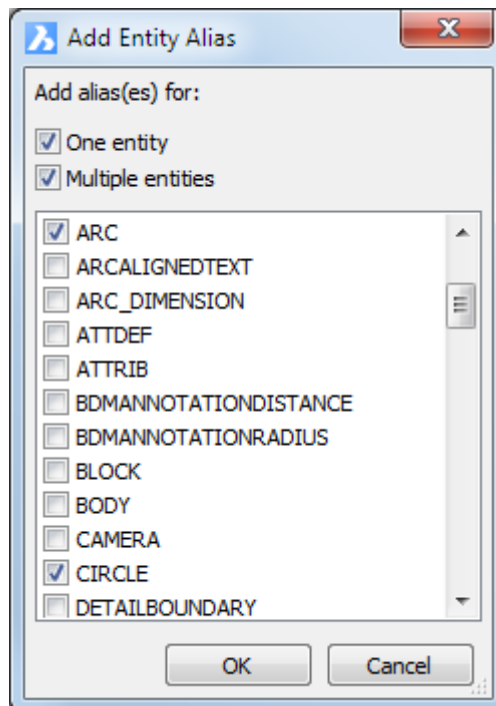
4. (option) Add a Quad Item Group:
 - Do one of the following:
 - Right click the *Custom* command group name, then choose *Append quad item group* in the context menu to add a new group at the bottom of the list.
 - Right click a group item name then choose *Append quad item group* in the context menu to add a new group above the selected item group.

The *Add Quad Item Group* dialog box displays.

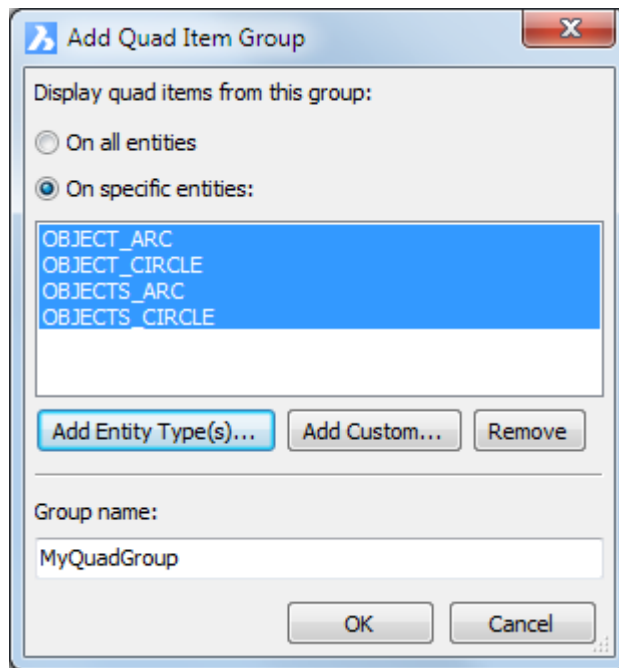
- Type a name in the *Group name* field.



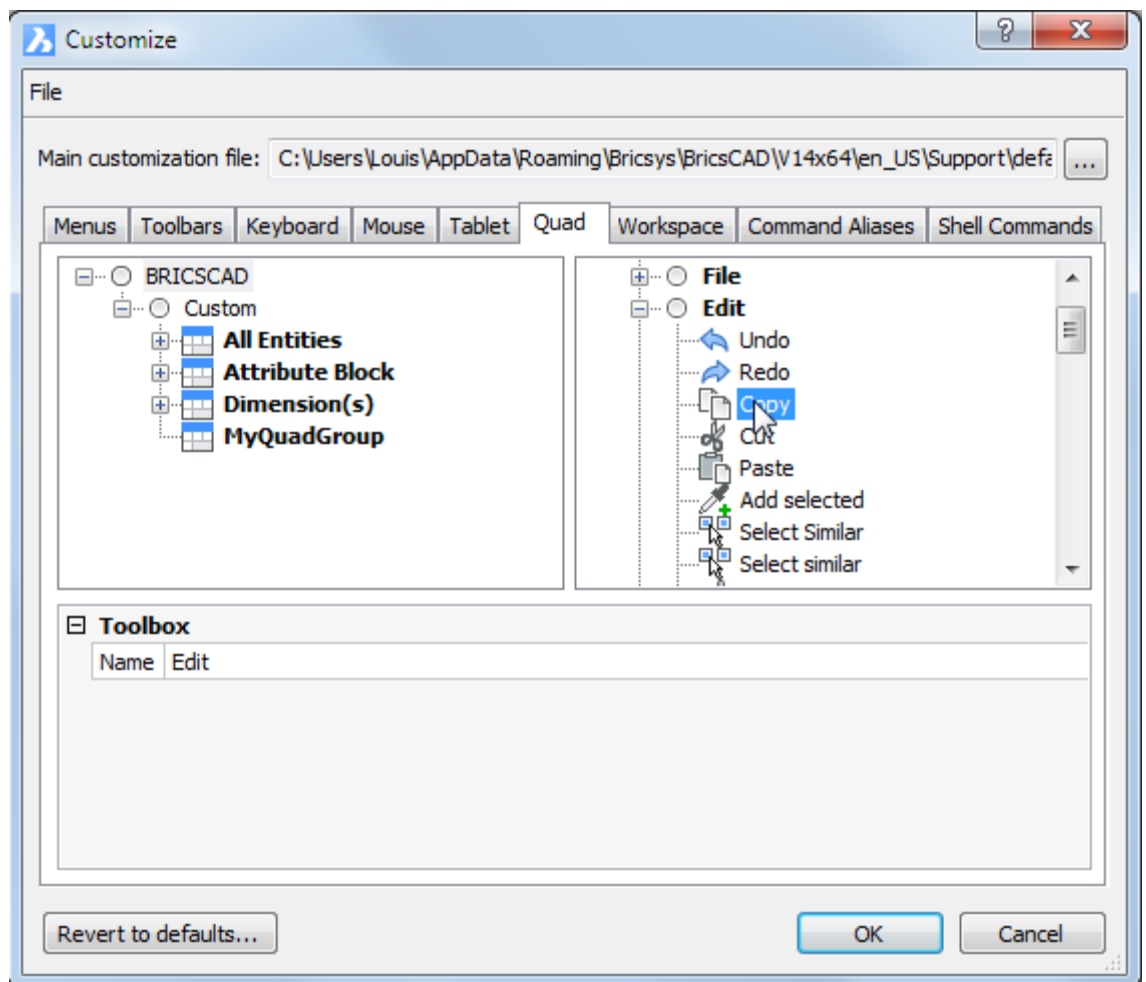
- Do one of the following:
- Select *On all entities* to apply the new group to all entity types, and click the *OK* button to create the new item group.
- Select *On Specific entities* to apply the new group to one or more specific entity types, and proceed with the next step.
- Click the *Entity Type(s)...* button and select one or more entity types.



- Check either One entity or Multiple entities or both.
- Check one or more entity types in the list.
- Click the *OK* button.



5. Click the *OK* button to create the new item group.
6. (option) Add a command to an item group:
Select a tool in the right hand pane of the *Customize* dialog box, then drag the tool to the item group.



7. Click the *OK* button on the *Customize* dialog to save the changes.





2D Constraints

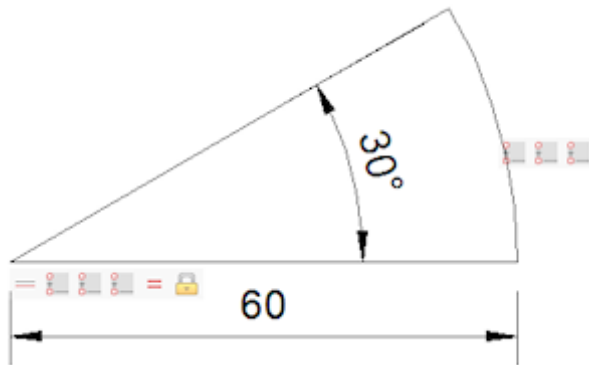
Parametric constraints are used to maintain relationships and control dimensions of 2D geometry.

There are two types of constraints:

- *Geometrical* constraints control relationships between entities.
- *Dimensional* constraints are used to control the dimensions of an entity, such as length, angle, radius or diameter.

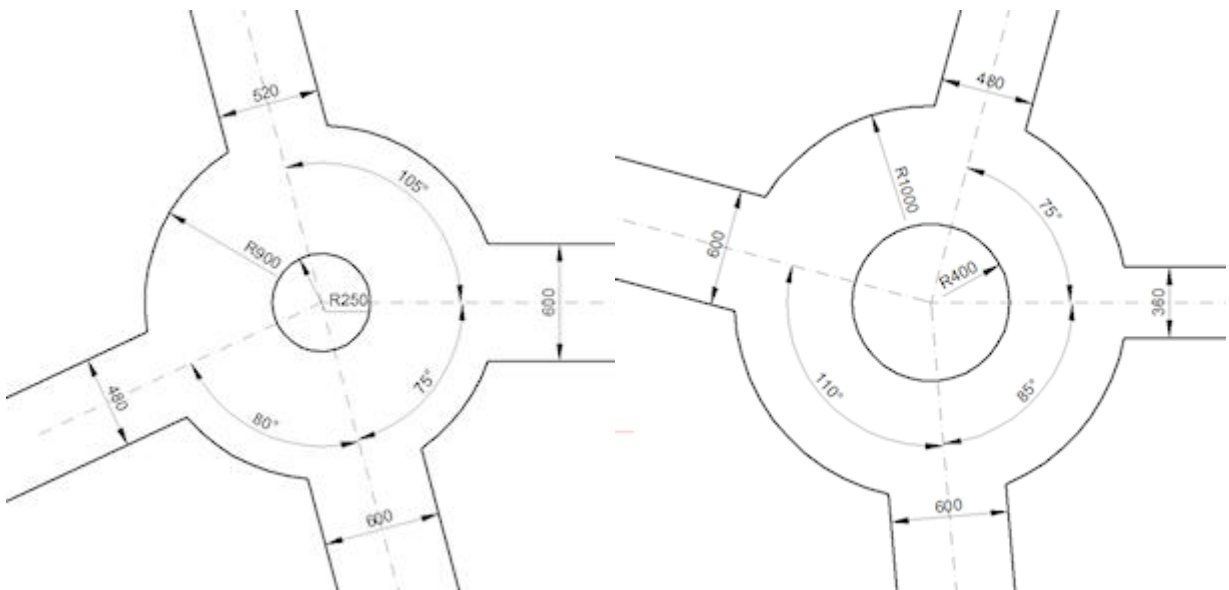
In the example below:

- The width and opening angle are controlled by dimensional constraints.
- Coincident geometrical constraints () are applied to the endpoints of the arc and the polyline and to the center point of the arc and the polyline vertex.
- The polyline segments have an equal () geometrical constraint.
- One segment of the polyline has a horizontal () geometrical constraint.
- The polyline vertex has a fix () geometrical constraint.



Parametric constraints allow to:

- Maintain geometric design intent.
- Make multiple versions of a design by applying different values to dimensional constraints.



Two versions of the same fully constrained drawing representing a roundabout

NOTE It is recommended to first apply geometric constraints, then dimensional constraints.

When using constraints a drawing can be in one of three states:

- Unconstrained: no entities are constrained. The drawing can be modified using modification commands and procedures.
- Underconstrained: some entities are constrained. It might be impossible to apply some changes using modification commands and procedures.
- Fully constrained: all possible and relevant geometric and dimensional constraints are applied to the design geometry. The drawing can be modified only by changing the values of the dimensional constraints.



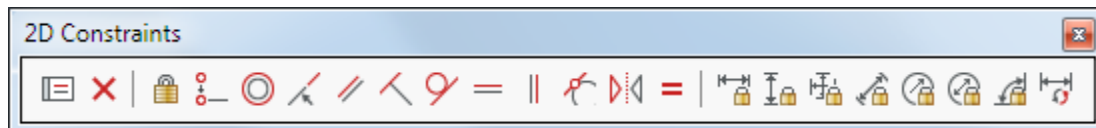
Door Swing (Internet connection needed)














Bracket (Internet connection needed)

Commands and toolbars

2D parametric constraints tools sit on the *2D Constraints* toolbar:



Command	Icon	Description
ConstraintBar		Controls the display of the constraint bar of entities. Constraint bars are always hidden when opening a drawing.
Delconstraint		Removes all dimensional and geometrical constraints from an entity.
Geometric Constraints		
GeomConstraint		Applies geometric relationships between entities and/or valid constraint points on entities.
GcFix		Constrains points and entities at a fixed position.
GcCoincident		Applies a coincident geometrical constraint to two points or constrains a point to an entity.
GcConcentric		Constrains the center points of circles, arcs, ellipses or elliptical arcs to coincide.
GcCollinear		Forces lines or polyline segments to be collinear.
GcParallel		Forces two lines or linear polyline segments to be parallel to each other.
GcPerpendicular		Constrains two lines or polyline segments to lie perpendicularly to each other.
GcTangent		Constrains two entities to maintain a point of tangency to each other or their extensions. The following entities are accepted: lines, polyline segments, circles, arcs, ellipses or elliptical arcs.
GcHorizontal		Constrains lines, linear polyline segments or pairs of points to lie parallel to the X-axis of the current coordinate system.
GcVertical		Constrains lines, linear polyline segments or pairs of points to lie parallel to the Y-axis of the current coordinate system.

GcSmooth		Forces a spline to maintain fluid geometric continuity with another spline, line, arc or polyline.
GcSymmetric		Constrains two entities or points to lie symmetrically with respect to a selected line.
GcEqual		Constrains arcs and circles to the same radius, or lines and polyline segments to the same length.
Dimensional Constraints		
Dimconstraint		Applies a dimensional constraint to an entity or between constraint points on entities. Converts associative dimensions to dynamic dimensions.
DcHorizontal		Constrains the horizontal distance (X-distance) between two points with respect to the current coordinate system.
DcVertical		Constrains the vertical distance (Y-distance) between two points with respect to the current coordinate system.
DcLinear		Constrains the horizontal distance (X-distance) or vertical distance (Y-distance) between two points with respect to the current coordinate system.
DcAligned		Constrains the distance between two points or the length of a line or polyline segment.
DcRadius		Constrains the radius of a circle, an arc or an arc polyline segment.
DcDiameter		Constrains the diameter of a circle, an arc or an arc polyline segment.
DcAngular		Constrains the angle between two lines or linear polyline segments; the total angle of an arc or an arc polyline segment; or the angle between three points on entities.
DcConvert		Converts an associative dimension to a dimensional constraint.

NOTE The PARAMETERCOPYMODE system variable controls copying of constraints and related parameters when entities are copied.

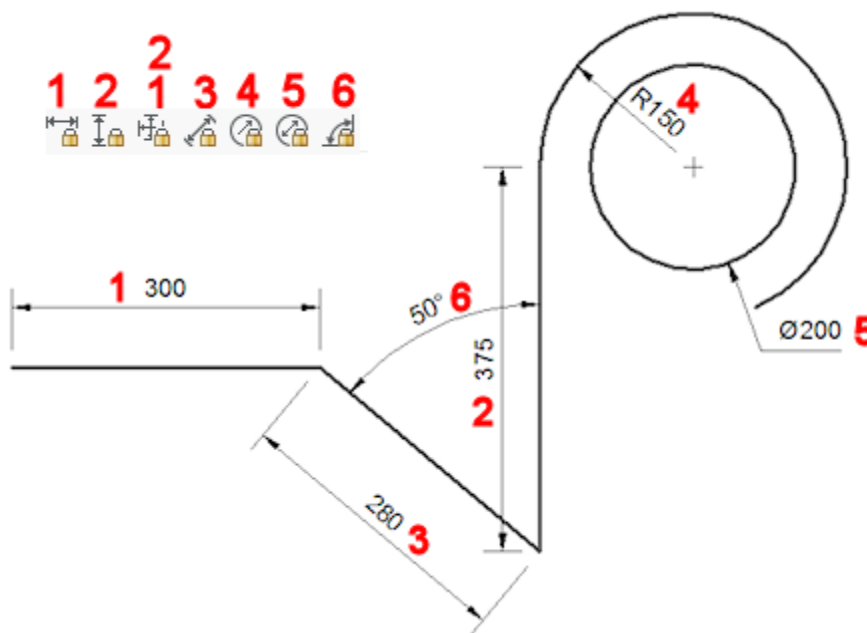
- 0: Do not copy 2D constraints.
- 1: Replace all expressions with constants.
- 2: Use existing parameters; if a parameter is missing, replace it with a constant.
- 3: Use existing parameters; create a new parameter if a parameter is missing.
- 4: Use existing parameters; create a new parameter if a parameter is missing, or has a different value.

Using Dimensional Constraints

Commands: DIMCONSTRAINT, DCALIGNED, DCANGULAR, DCCONVERT, DCDIAMETER, DCHORIZONTAL, DCLINEAR, DCRADIUS, DCVERTICAL, DELCONSTRAINT

The dimensions of the geometry can be controlled by dimensional constraints:

- Horizontal distance between points (1).
- Vertical distance between points (2).
- Distance between points or the length of lines and polyline segments (3).
- Radius of arcs, circles and arc polyline segments (4).
- Diameter of arcs, circles and arc polyline segments (5).
- Angle between entities or between points and entities (6).



NOTE Dimensional constraints are created using the current dimension style.

To edit dimensional constraints

Dimensional constraints can be edited:

- using the DDEDIT command
- in the Properties bar
- in the Mechanical Browser

To edit dimensional constraints in the Properties bar.

1. Select the dimensional constraint.
The properties of the entity display in the *Properties* bar.
2. Type a new value or expression in the *Expression* field, then press Enter.
The readout of the *Value* field changes accordingly.

Constraint	
Constraint Form	Dynamic
Reference	No
Name	d1
Expression	
Value	200.0
Description	Length horizontal leg
Text	
Text rotation	0.0000

3. (option) Edit the *Name* field.
4. (option) Edit the *Description* field.
5. (option) Edit the *Text rotation* field.
6. Press the Esc key to unselect.

NOTE Select multiple dimensional constraints simultaneously to set their value equally.

To edit dimensional constraints in the Mechanical Browser:

1. If not already open, open the Mechanical Browser.

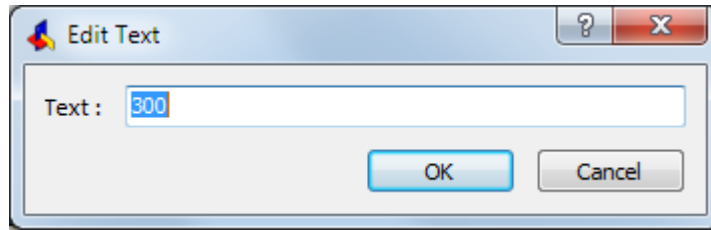
The screenshot shows the Mechanical Browser window with the 'Library' tab selected. The tree view shows a 'Bracket' model containing a 'Parameters' folder. Inside 'Parameters', there are four parameters: 'd1 = 200', 'd2 = 150', 'dia1 = 50', and 'dia2 = dia1*0.5'. The bottom panel, titled 'Parameter', displays the properties for the selected parameter 'd1':

Parameter	
Name	d1
Expression	200
Value	200
Description	Length horizontal leg

2. Select a parameter.
The properties the selected parameter display in bottom part of the *Mechanical Browser*.
3. (option) Select the *Name* field, then type a new name.
4. (option) Select the *Expression* field, then type a new value or expression.
5. (option) Select the *Description* field, then type a new description.

To edit a single dimensional constraint:

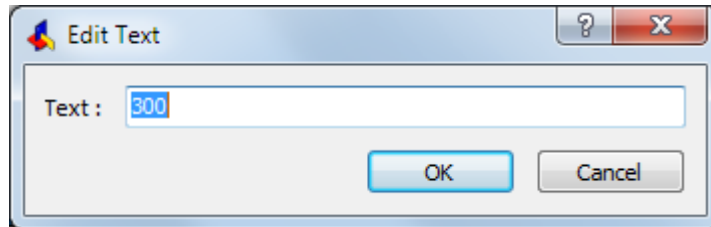
1. Double click the dimensional constraint.
The command bar reads: `_ddedit`
The *Edit Text* dialog box displays.



2. Type a new value in the *Text* field, then press Enter or click the *OK* button.
The new value is applied and the geometry is rebuild accordingly.

To edit multiple dimensional constraints:

1. Type `ddedit` or `ed` in the command bar, then press Enter.
The command bar reads: Select entity to modify:
2. Click the a dimensional constraint.
The *Edit Text* dialog box displays.



3. Type a new value in the *Text* field, then press Enter or click the *OK* button.
4. (option) Repeat steps 2 and 3 to edit more dimensional constraints.
5. Press Enter or right click to stop editing dimensional constraints.
All edited dimensional constraints are updated simultaneously.
The geometry is rebuild accordingly.

Using expressions to define dimensional constraints

Each dimensional constraint is named automatically upon its creation.

The naming is as follows:

distance: `d1`, `d2`, `d3`, ...

radius: `rad1`, `rad2`, `rad3`, ...

diameter: `dia1`, `dia2`, `dia3`, ..

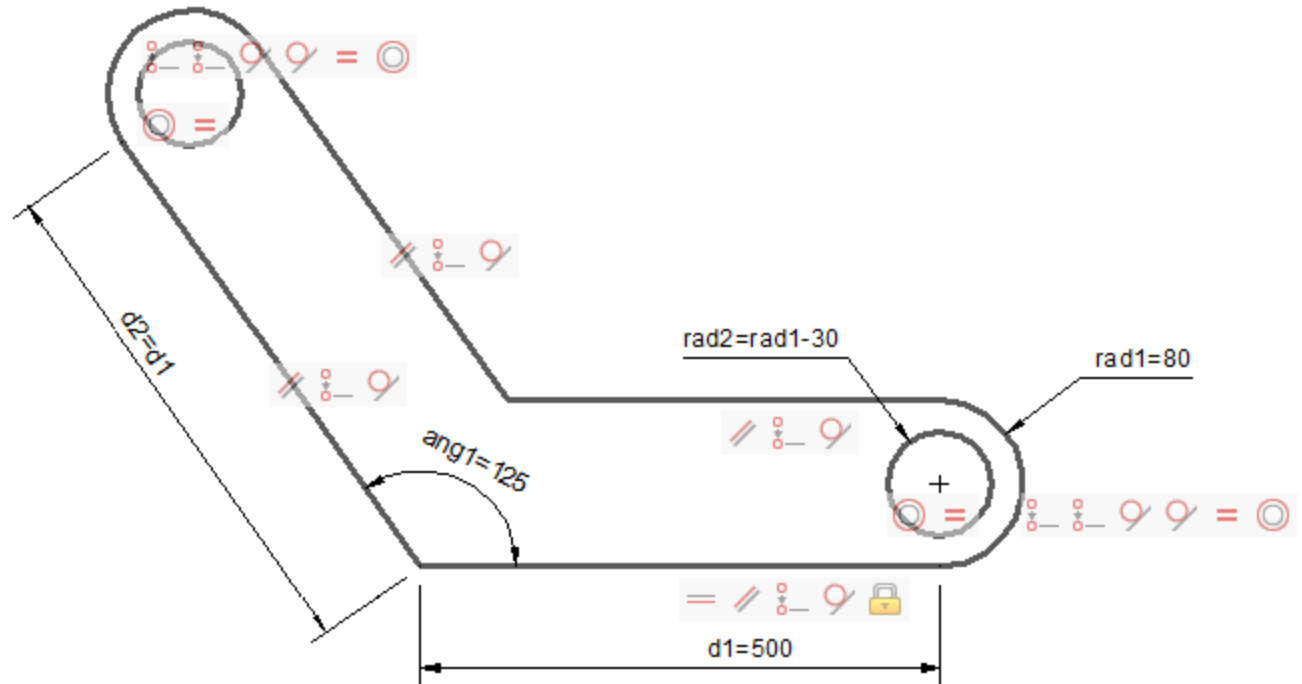
angular: `ang1`, `ang2`, `ang3`, ..

The name of a constraint can be used as a parameter in a mathematical expression to define another dimensional constraint.

The dimensions of the bracket below are controlled through the *rad1* radius, *d1* distance and *ang1* angular dimensional constraints.

The radius of the hole (*rad2*) and the length of the second leg (*d2*) are defined by expressions.

Geometrical constraints force the width of the second leg and the hole in it to be equal to the first leg.



Creating a new parameter

1. Select either the part name or *Parameters* in the *Mechanical Browser*, then right click.
A context menu displays.
2. Select *Add new parameter* in the context menu.
A new parameter is created.
3. Edit the properties of the new parameter.

Parameter	
Name	v1
Expression	1.0
Value	1
Description	

- NOTES**
- The constants *Pi* and *e* can be used in expressions.
 - The CLEANUNUSEDVARIABLES command purges parameters that are not used in constraint expressions or linked to dimensions.

Using operators in expressions

The following operators can be used in expressions:

Operator	Description
+	Addition
-	Subtraction or Negative
*	Multiplication
/	Division
^	Exponentiation
%	Modulo or Remainder operator The expression "5 % 2" would evaluate to 1, because 5 divided by 2 leaves a quotient of 2 and a remainder of 1 .

Expressions are evaluated according to the standard mathematical rules of precedence:

1. Expressions within brackets; innermost sets first.
2. Standard operations order:
3. exponent
4. multiplication and division
5. addition and subtraction
6. Operators of equal precedence from left to right.

Using functions in expressions

The following functions can be used in expressions:

Function	Syntax
Cosine	cos(expression)
Sine	sin(expression)
Tangent	tan(expression)
Arc cosine	acos(expression)
Arc sine	asin(expression)
Arc tangent	atan(expression)
Hyperbolic cosine	cosh(expression)
Hyperbolic sine	sinh(expression)
Hyperbolic tangent	tanh(expression)
Arc hyperbolic cosine	acosh(expression)
Arc hyperbolic sine	asinh(expression)
Arc hyperbolic tangent	atanh(expression)
Square root	sqrt(expression)

Signum function (-1,0,1)	sign(expression)
Round to nearest integer	round(expression)
Truncate decimal	trunc(expression)
Round down	floor(expression)
Round up	ceil(expression)
Absolute value	abs(expression)
Largest element in array	max(expression1;expression2) ¹
Smallest element in array	min(expression1;expression2) ¹
Degrees to radians	d2r(expression)
Radians to degrees	r2d(expression)
Logarithm, base e	ln(expression)
Logarithm, base 10	log(expression)
Exponent, base e	exp(expression)
Exponent, base 10	exp10(expression)
Power function	pow(expression1;expression2) ¹
Random decimal, 0-1	Random

(1) Use the list separator character as defined on your system: , (comma) or ; (semicolon).

Using Geometric Constraints

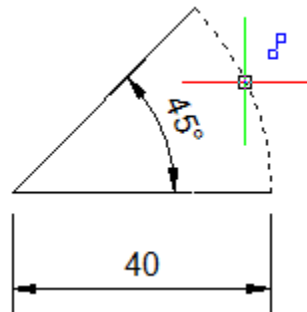
A geometric constraint is a non-numerical relationship between the parts of a geometric figure.

Geometric constraints:

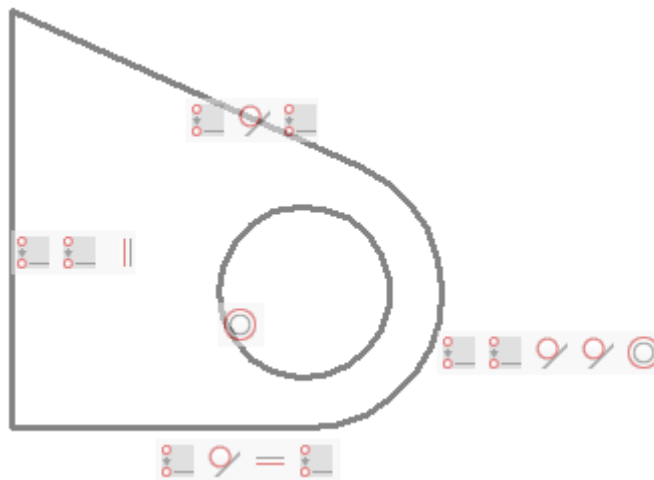
- associate geometric entities together two by two (coincident, concentric, collinear, parallel, perpendicular, tangent, smooth, symmetric, equal)
- specify a fixed angle (horizontal, vertical)
- specify a fixed location (fix)

When a geometric constrained is applied to an entity:


- the position of the entity is adjusted according to the applied constraint.
- an icon displays next to the entity, to indicate the applied constraint. If multiple geometric constraints are applied, the icons are joined to a *constraint bar*.
- If the `SELECTIONPREVIEW` system variable is set to 1 or 3, a blue icon displays next to the cursor when you move the cursor over entities that have constraints applied to them.



In the image below the endpoints of the three lines and the arc are joined by coincident constraints. The midpoint of the circle and the arc are made concentric. Two lines are forced to be tangent to the arc. One line has a vertical constraint (= parallel to the Y-axis of the current coordinate system), one line has a horizontal constraint (= parallel to the X-axis of the current coordinate system)



To control the display of constraint bars

1. Click the *2D Constraint bar* tool button () on the *2D Constraints* toolbar.
The command bar reads: Select option to [Show/Hide/Reset] constraints:

A prompt menu displays:



2. Do one of the following:
 - Choose *Hide* or *Show* in the context menu.
 - Type *H* or *S* in the command bar, then press Enter.

The command bar reads: Select entities:

3. Select entities, then press Enter.

NOTE Constraint bars are always hidden when opening a drawing.


To control the position of a constraint bar

By default constraint glyph bars are created close to the midpoint of the entity and are kept at that relative position when the entity position changes. You can drag the constraint bar to a different location. This new relative position is then maintained until the *Reset* option of the CONSTRAINTBAR command restores the default position of the constraint bar.

To relocate a constraint bar:

1. Place the cursor on the constraint bar.
2. Press and hold the left mouse button to move the constraint bar.
3. Release the left mouse button at the desired location.

To restore the default position of the constraint bars:

1. Click the *2D Constraint bar* tool button () on the *2D Constraints* toolbar.
The command bar reads: Select option to [Show/Hide/Reset] constraints:

A prompt menu displays:



2. Do one of the following:
 - Choose *Reset* in the prompt menu.
 - Type *R* in the command bar, then press Enter.

The command bar reads: Select entities:

3. Select entities, then press Enter.

Working with the constraint bar

Using the constraint bar you can:

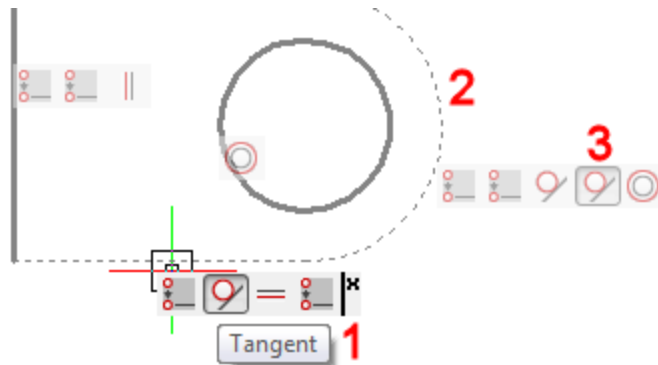
- Control a constraint.

Move the cursor over a constraint icon.

A tooltip displays, indicating the constraint type (1)

The associated entity highlights (2)

The corresponding icon on the constraint bar of the associated entity highlights (3)

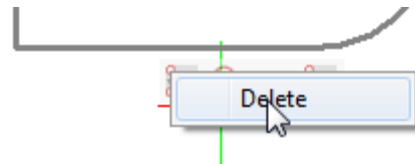


- Delete a constraint.

Move the cursor over the constraint icon, then right click.

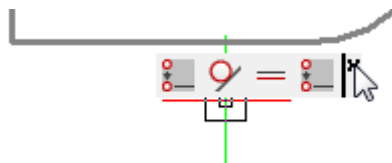
Click *Delete*.

The constraint is deleted and the icon is removed from this constraint bar and from the constraint bar of the corresponding entity.



- Hide the constraint bar of an entity.

Move the cursor of the constraint bar, then click the *Close* button.



Deleting constraints

To delete a single constraint: right click the constraint icon, then click the *Delete* button.

To delete all constraints from a selection set:

1. Click the *Delete 2D Constraints* icon (X) on the *2D Constraints* toolbar.
The command bar reads: Select entities to delete the constraints from:
2. Select the entities you want to delete the constraints from.
3. Right click to stop selecting entities and delete all constraints from the selection set.

Using fix constraints

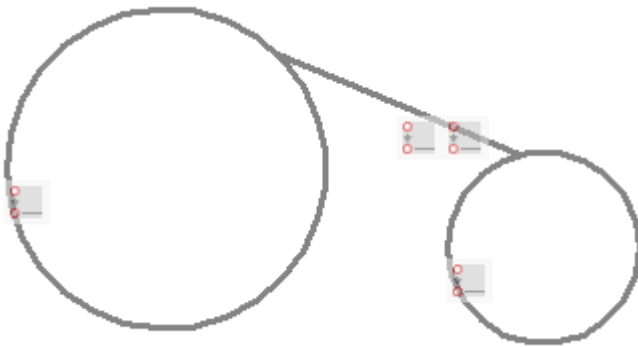
A fix geometric constraint (🔒) forces a point or an entity to a fixed location.

- If a fix geometric constraint is applied to a line or a polyline segment, the angle of the line or polyline segment is fixed. The endpoints can still be moved.
- If you fix an arc or a circle, the center point is fixed too.

NOTE It is recommended to apply a fix constraint to important geometric features in your design. This prevents the geometry from moving unexpectedly when you edit the design.

Using coincident constraints

A coincident geometric constraint can be applied to two points or to a point and an entity. The endpoints of lines, polylines, splines, rays and arcs (circular and elliptical) and the center point of circles, ellipses and arcs (circular and elliptical) are accepted. These points can be made coincident (= to lie on the entity or on the extension of the entity) with a line, a polyline segment, a circle or an arc (circular and elliptical).



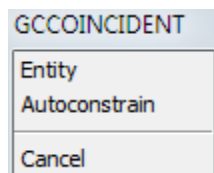
Coincident constraints between the endpoint of a line and two circles

If the endpoints or center points of entities already coincide, the *Autoconstraint* option of the *GcCoincident* command automatically applies coincident constraints to such points.

To apply a coincident constraint between a point and an entity:

1. Click the *Coincident* tool button (📍—) on the *2D Constraints* toolbar. The command bar reads: Select first point or [Entity/Autoconstrain] <Entity>:

A prompt menu displays:



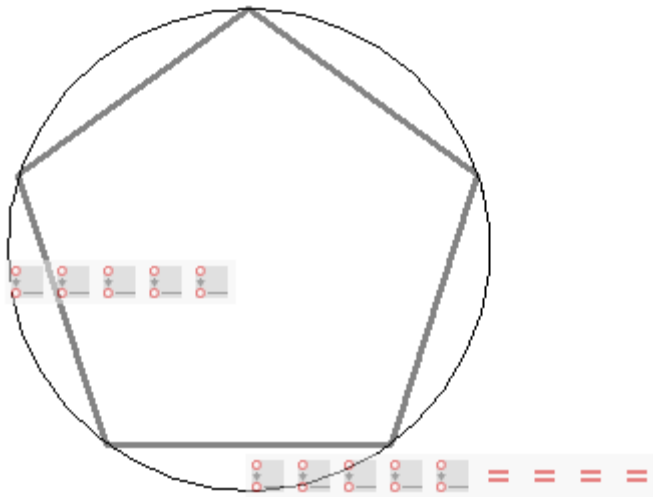
2. Do one of the following:
 - Press Enter.
 - Choose *Entity* in the prompt menu.

The command bar reads: Select an entity:

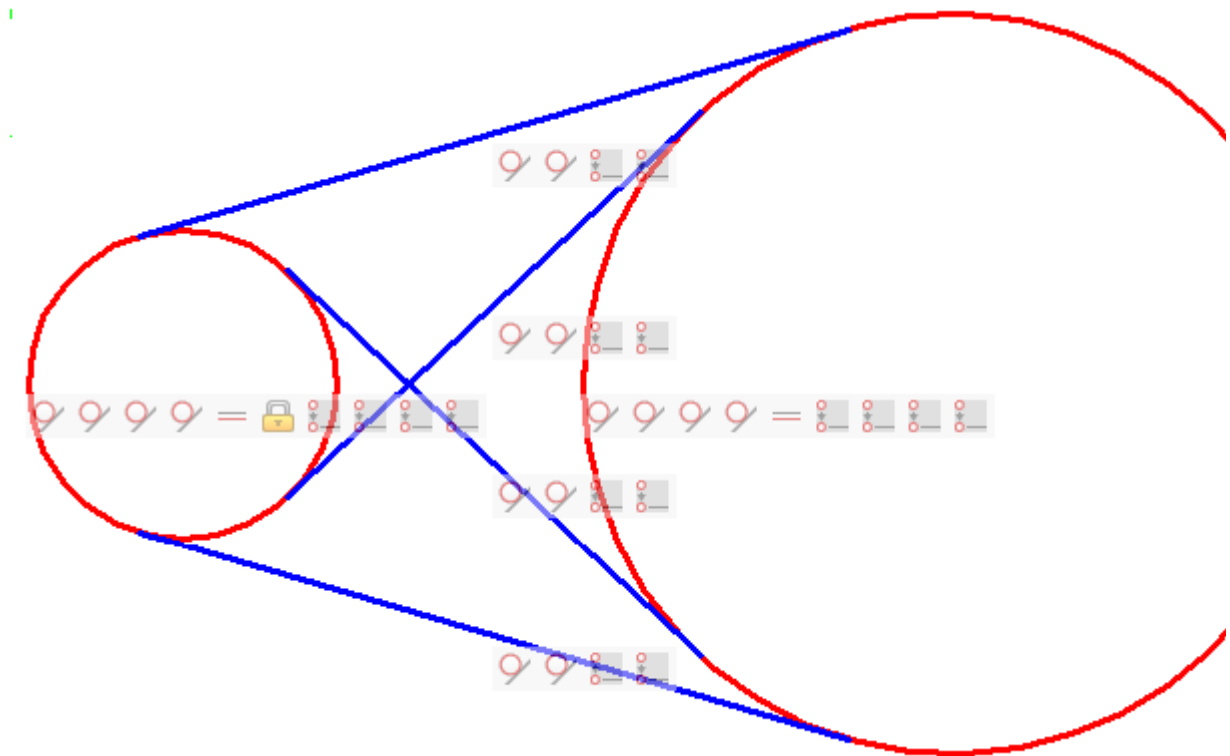
3. Select the entity.
The entity highlights.
The command bar reads: Select point or [Multiple]:
4. Snap to the endpoint of an entity or the center point of a circle or an arc.
The selected point is forced to lie on the selected entity (or its extension).

NOTE If you choose *Multiple* in step 3, you can select multiple points.

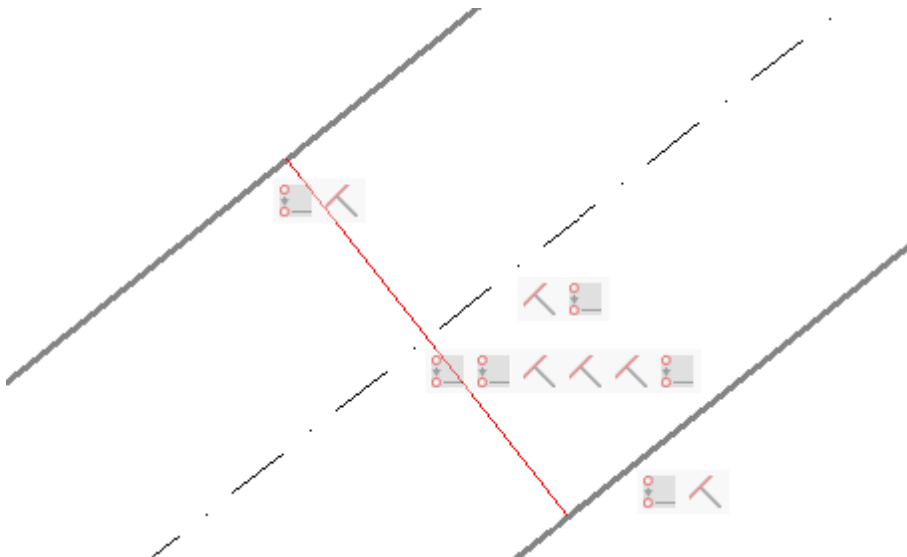
Usage of a coincident constraint between a point and an entity



By applying coincident constraints between the pentagon vertices and the circle, and equal constraints between one side and the four other sides, the circle radius defines the size of the pentagon.



Coincident constraints between the endpoints of the tangent lines and the circles prevent the tangent lines to extend beyond the tangent points.



Coincident constraints are used to:

- force the endpoints of the red line to lie on the bold lines;
- force the midpoint of the red line to lie on the dash-dot line.


The bold lines and the dash-dot line have a perpendicular constraint with the red line.

As a result the dash dot line will always lie in the middle of the two bold lines.

Using horizontal and vertical constraints

Horizontal and vertical geometric constraints force two points, a line or a polyline segment to be parallel to the X-axis (horizontal) or the Y-axis (vertical) of the current coordinate system. Therefore these constraints can be used to keep lines at a fixed angle, rather than only horizontal or vertical.

To constrain a line to a fixed angle

1. Align the X-axis of the coordinate system to the desired angle.
 - Type *UCS* in the command bar, then press Enter.
 - Choose *Z* in the prompt menu or type *Z*, then press Enter.
 - Type the angle, then press Enter.
2. Click the *Horizontal* tool button () on the *2D Constraints* toolbar.
The command bar reads: Select an entity or [2Points] <2Points>:
3. Click the line or polyline segment.
The selected entity is constrained to be parallel to the X-axis of the current UCS.
4. (option) Restore the WCS: Type *UCS* in the command bar, then press Enter twice.

NOTE Use the *Vertical* tool in the above procedure to constrain entities to be parallel to the Y-axis of the current UCS.

Dynamic Dimensions

Dynamic dimensions provide an interface at the cursor position to specify the length and angle of the linear entity being created or grip-edited. As such dynamic dimensions help you to focus on the drawing area.

The behavior and appearance of dynamic dimensions is controlled through a series of system variables and user preferences.

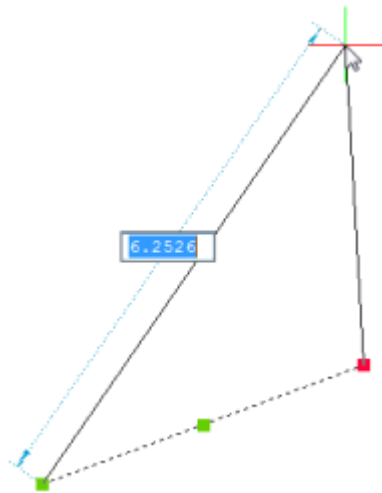
Defining the Dynamic Dimensions Settings

1. Right click the *DYNMODE* field in the Status Bar, then choose *Settings* in the context menu.

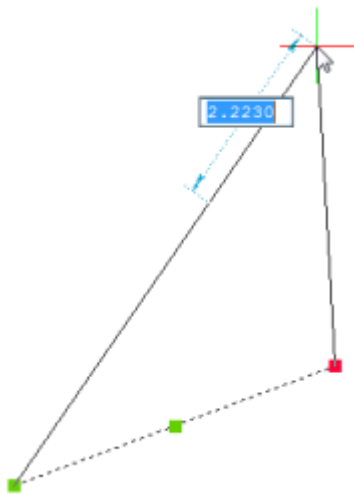
The *Dynamic input* settings in the Settings dialog display:

Drawing	
Drafting	
+ Drawing units	
+ Coordinate input	
Dynamic input	
Dynamic input mode	0x0003 (3)
negative	<input type="checkbox"/> Switch all off temporarily
0x0001	<input checked="" type="checkbox"/> Dynamic input at pointer (not supported)
0x0002	<input checked="" type="checkbox"/> Editable dynamic dimensions
Show dynamic dimensions	0x001F (31)
0x0001	<input checked="" type="checkbox"/> Resulting length
0x0002	<input checked="" type="checkbox"/> Extended length
0x0004	<input checked="" type="checkbox"/> Absolute angle
0x0008	<input checked="" type="checkbox"/> Relative angle
0x0010	<input checked="" type="checkbox"/> Arc radius
Dynamic dimension visibility	Only the first dynamic dimension
Dynamic dimension hover color	<input checked="" type="checkbox"/> 142
Dynamic dimension hot color	<input checked="" type="checkbox"/> 142
Dynamic dimension distance	1
Dynamic dimension linetype	Dotted

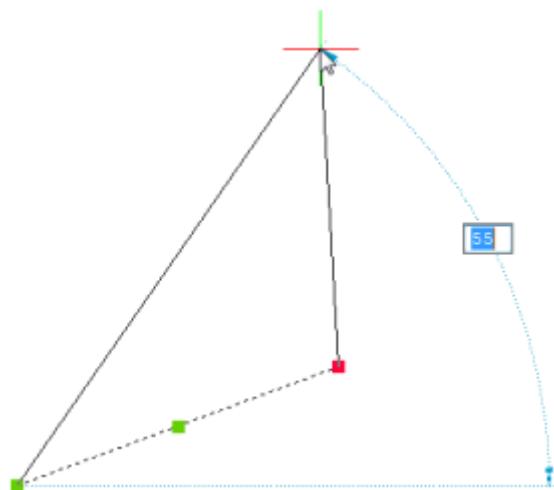
2. Define the *DYNMODE* system variable, to switch the display of dynamic dimensions and/or dynamic input on/off.
Please notice that dynamic input at the pointer is not implemented yet.
3. Define the *DYNDIGRIP* system variable, to select the dynamic dimension types to be displayed.
 - Resulting length: total length of the line or polyline segment



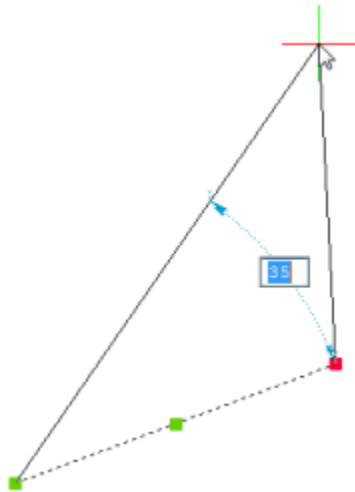
- Extended length: incremental length of the line



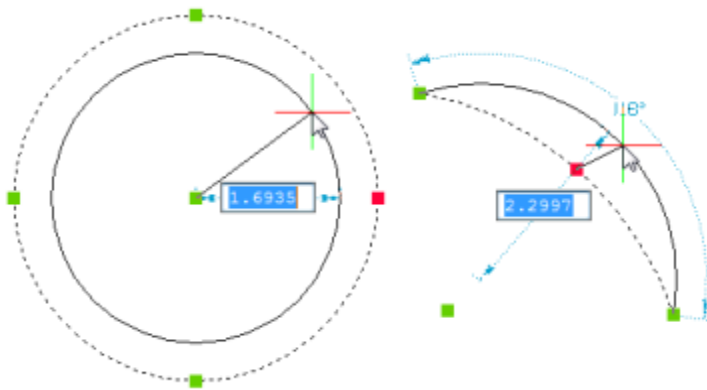
- Absolute angle: angle relative to the X-axis of the current UCS



- Relative angle: angle relative to the original angle of the linear entity



- Arc radius: radius of arcs and circles



4. Define the *DYNDIVIS* system variable to choose the number of dynamic dimensions that display simultaneously.
 - *Only the first dynamic dimension (DYNDIVIS=0)*: hit the TAB key to display and edit the next dynamic dimension, as defined by the *DYNDIGRIP* system variable.
 - *Only the first two dynamic dimension (DYNDIVIS=1)*: hit the TAB key to edit the next dynamic dimension, as defined by the *DYNDIGRIP* system variable and display the next plus one.
 - *All dynamic dimensions as defined by DYNDIGRIP (DYNDIVIS=2)*: hit the TAB key to edit the next dynamic dimension.
5. Set the dynamic dimensions user preferences to define the appearance (color, linetype and position) of dynamic dimensions.

Using dynamic dimensions when creating entities

1. Launch the command to create a 2D entity (e.g. Line and *DYNDIVIS* = 1 or 2).
2. Specify the first point, then move the cursor to display the dynamic dimensions. The *Length* dynamic dimension highlights.
3. Do one of the following:
 - Type the desired length of the line, then hit the TAB key to jump to the *Angle* dynamic dimension.
The *Length* dynamic dimension is locked and displays in red and the *Angle* dynamic dimension is selected.
 - Type the desired length of the line, then click to create the line at the angle defined by the cursor position.
 - Hit the TAB key to jump to the *Angle* dynamic dimension.

If *DYNDIVIS* = 0, the *Length* dynamic dimension disappears when you hit the TAB key.

4. Do one of the following:
 - Type the desired angle, then hit the Enter key to create the line.
 - Type the desired angle, then hit the TAB key to return to the *Length* dynamic dimension.
 - Hit the Enter key to create the line at the angle defined by the cursor position.
 - Hit the TAB key to return to the *Length* dynamic dimension.

If *DYNDIVIS* = 0, the *Arc* dynamic dimension disappears when you hit the TAB key.

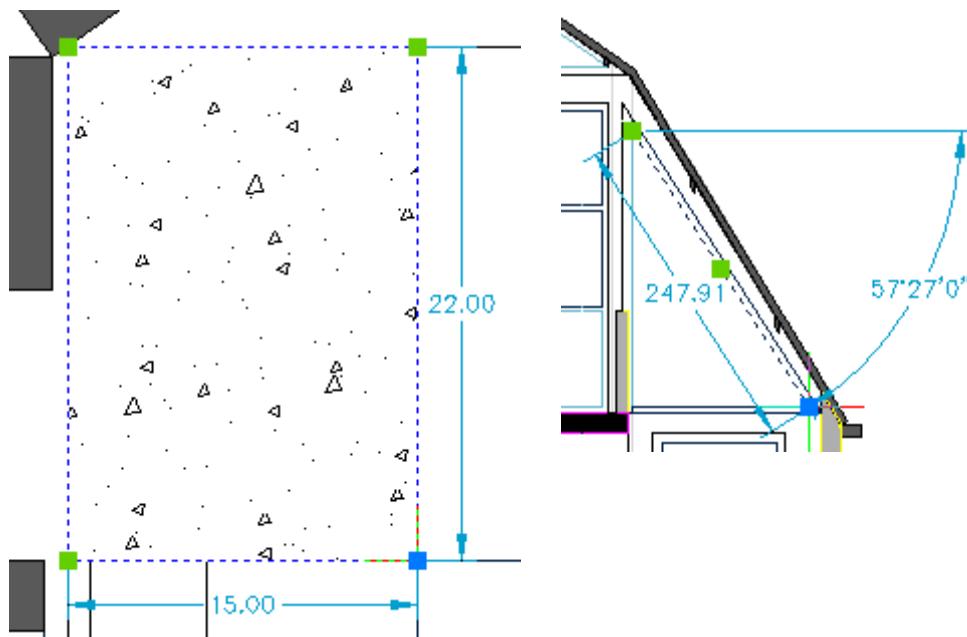
NOTE If ORTHO is on, you can still specify the angle using the *Angle* dynamic dimension.

Using dynamic dimensions to grip-edit entities

1. Select the entity or entities.
All grips on the selected entities display.
2. Click the grip you want to edit, then move the cursor.
Depending on the value of the *DYNDIVIS* system variable one, two or all dynamic dimensions display dynamically.
The first dynamic dimension highlights.
3. Do one of the following:
 - Type the desired value for the currently highlighted dynamic dimension, then hit the TAB key to jump to the next dynamic dimension.
The dynamic dimension is locked and displays in red.
 - Type the desired value for the currently highlighted dynamic dimension, then press Enter to stop.
 - Hit the TAB key to jump to the next dynamic dimension.
4. Repeat step 3 or press Enter to stop.

Using dynamic dimensions to measure entities

1. Select the entity or entities.
All grips on the selected entities display.
2. Hover the cursor over a grip.
The grip highlights and all dimensions related to the selected grip display.



Dynamic dimensions used to measure a rectangle (left) and a line (right).

3. (option) Click to start editing the selected grip.

Coordinate Input

When you create entities in a drawing, they are located in relation to the drawing's underlying Cartesian coordinate system. Every drawing has a fixed coordinate system called the *World Coordinate System* (WCS).

You can also define arbitrary coordinate systems located anywhere in three-dimensional space. These are called user coordinate systems (UCS) and can be located anywhere in the WCS and oriented in any direction.

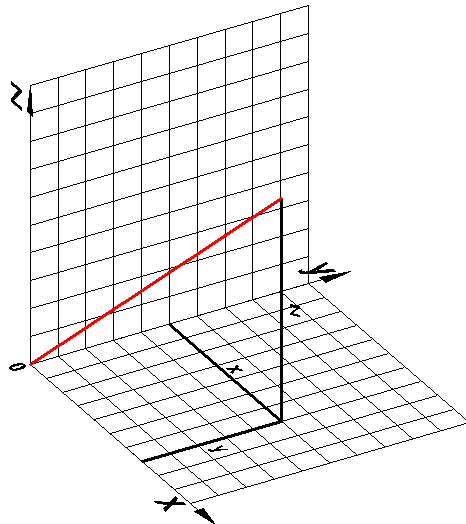
To specify points and distances using the keyboard you can use the following formats:

- Cartesian coordinates: x,y,z
- Cylindrical coordinates: $R<\alpha,z$
- Spherical coordinates: $R<\alpha<\beta$

If you place the @-character in front of the entry, the coordinates are calculated with respect to the previous point. This technique is called: *Relative Coordinates*.

Working with Cartesian coordinates

In the Cartesian coordinate system we use three perpendicular axes: the x-axis, the y-axis and the z-axis. All axes originate in the origin point of the coordinate system. The x-axis and the y-axis define a horizontal plane, while the x-axis and the z-axis and the y-axis and the z-axis define vertical planes. A point is defined by its distances to the yz-, xz- and xy- planes. These distances are called the xyz-coordinates of a point.



If you want to enter the absolute Cartesian coordinates of a point, type the x-, y- and z-coordinates separated by commas: 45.5,57.3,60

If you omit the z-coordinate, the point is placed in the xy-plane ($Z = 0$).

If you place the @-character in front of the entry (@x,y), the coordinates are calculated with respect to the previous point. This technique is called *Relative Cartesian coordinates*.

Using relative Cartesian coordinates to draw a rectangle

1. Launch the RECTANGLE command.
2. Specify the first corner of the rectangle.
3. In the command bar type: @<width>,<height>
 - <width> = the width of the rectangle in drawing units, measured along the x-axis
 - <height> = the height of the rectangle in drawing units, measured along the y-axis

Working with cylindrical coordinates

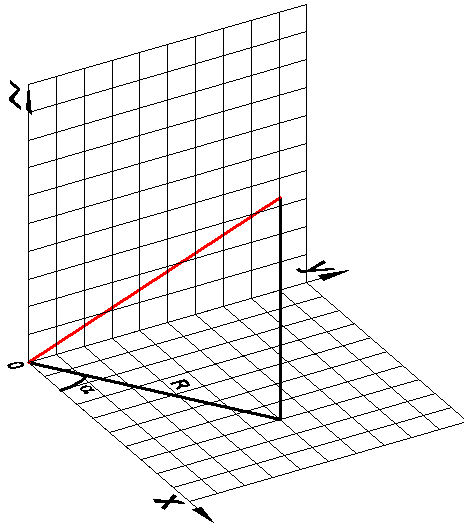
In a cylindrical coordinate system we use three perpendicular axes: the x-axis, the y-axis and the z-axis. All axes originate in the origin point of the coordinate system. The x-axis and the y-axis define a horizontal plane, while the x-axis and the z-axis and the y-axis and the z-axis define vertical planes.

A point is defined using the following format: R<alpha, z.

- R = distance to the origin in the xy-plane
- <alpha = the angle between R and the x-axis (positive angles are measured counter clockwise)
- z = the height above the xy-plane.

If the z-coordinate is omitted, cylindrical coordinates are referred to as polar coordinates.

If you place the @-character in front of the entry, the coordinates are calculated with respect to the previous point. This technique is called *Relative Cylindrical coordinates*.

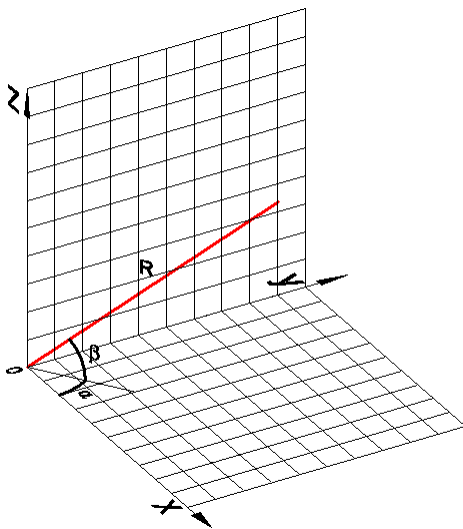


Working with spherical coordinates

In a cylindrical coordinate system we use three perpendicular axes: the x-axis, the y-axis and the z-axis. All axes originate in the origin point of the coordinate system. The x-axis and the y-axis define a horizontal plane, while the x-axis and the z-axis and the y-axis and the z-axis define vertical planes.

A point is defined using the following format: $R < \alpha < \beta$

- R = distance from the origin
- α = angle in the xy-plane (positive angles are measured counter clockwise)
- β = angle measured from the xy-plane (positive angles are measured counter clockwise, above the xy-plane)



Using Snap and Grid

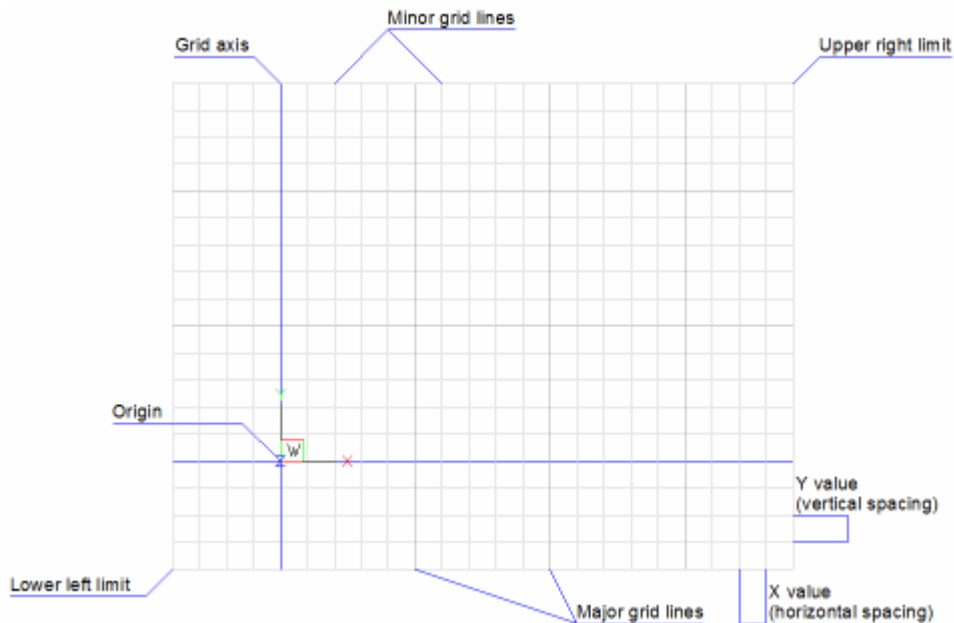
Commands: SNAP, GRID and LIMITS

Grid and snap help you to draw fast and accurately.



A *grid* is a set of evenly spaced lines or dots that serve as a visual distance reference.

The GRIDSTYLE system variable controls whether grid lines or dots display.

The GRIDDISPLAY system variable controls the display of the grid. The grid can also indicate how far the drawing limits extend.



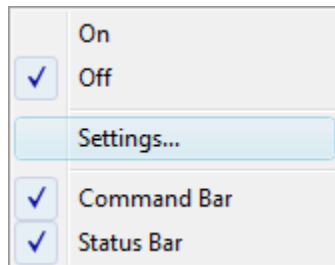
The *snap* feature creates a set of evenly spaced, invisible magnetic points, which make the crosshairs move in even increments. Both grid and snap are like the intersection points of the lines on a piece of grid paper. Grid points are for visual reference only and they do not print. *Snap* constrains the points that you can pick with the mouse.

- Click the *Grid* tool () on the *Settings* toolbar or the *Grid* field in the Status Bar, or press *Ctrl+G* to toggle grid display on/off.
- Click the *Snap* tool () on the *Settings* toolbar or the *Snap* field in the Status Bar, or press *Ctrl+B* to toggle snap on/off.
- Both *snap* and *grid* can be set differently in each viewport.
- The *Snap* option of the *Grid* command equals the grid spacing to the current snap spacing.
- Grid lines or dots do not print.

Display the Snap and Grid settings

Do one of the following:

- In the Status Bar, right click on the *Snap* field, then choose *Settings* in the context menu.



- Open the Settings dialog, then click the *Drawings* button. Under *Drafting*, expand *Coordinate Input* and *Snap/Grid*.

To synchronize snap and grid spacing


1. Type *Grid* in the command bar, then press Enter.
The command bar reads: Grid is on: OFF/Aspect/<Grid spacing (synced with snap)>:
2. Type *s*, then press Enter.
3. Type *Snap* in the command bar, then press Enter.
The command bar reads: Snap is on, x and y = 0.2000: OFF/Rotate/Style/Aspect/<Snap spacing>:
4. Type a new value for the snap spacing, then press Enter.
The value is applied to both snap and grid.

NOTE The grid spacing remains in sync with the snap spacing until you set the grid spacing differently using the Grid command or in the Settings dialog.

To set the Drawing Limits

See the LIMITS command.

To toggle the Drawing Limits

Click the *Drawing Limits* tool button () on the *Settings* toolbar to toggle the LIMCHECK system variable on/off.

If LIMCHECK = ON, it is not possible to draw outside the drawing limits.

The appearance (pressed or unpressed) of the *Drawing Limits* tool button reflects the current status of the *Drawing Limits*.

The *Display beyond LIMITS area* option of the GRIDDISPLAY system variable defines whether or not the grid display indicates the drawing limits.

Using Isometric Snap

You can use the Isometric snap and grid option to create two-dimensional isometric drawings. With the isometric option, you can draw a simulated three-dimensional view on a two-dimensional plane, much the same as you might draw on a piece of paper. Do not confuse isometric drawings with three-dimensional drawings.

The isometric option always uses one of three preset planes, which are denoted as *Left*, *Right* and *Top*. You cannot alter the arrangement of these planes. If the Snap Angle is 0, the three isometric axes are 30 degrees, 90 degrees, and 150 degrees.

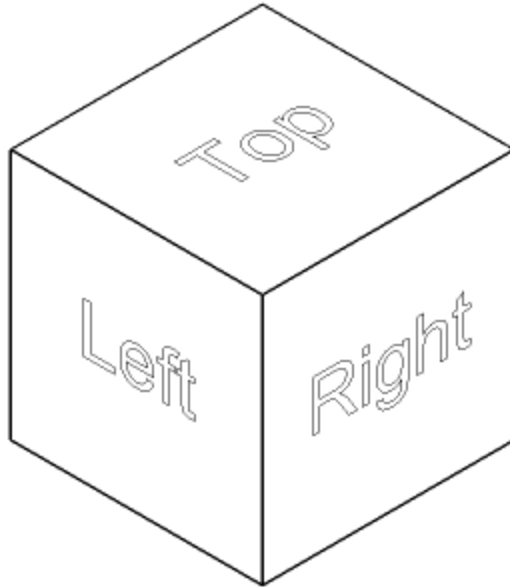
When you set the *Snap Style* setting to *Isometric Snap* and then set the *Snap Isometric Pair* setting to *Left*, *Top* or *Right*, the snap intervals, grid, and crosshairs align with the selected plane. The grid is always shown as isometric and uses y-coordinates to calculate the grid spacing. If the Orthogonal Mode is active, the movement of the crosshairs is constrained to the current isometric plane. Press the *F5* function key to toggle the *Snap Isometric Pair* setting.

1. Display the *Snap/Grid* settings in the Settings dialog:

Coordinate input	
Orthogonal mode	<input type="checkbox"/> Orthogonal mode
Tablet mode	[0] Command selection mode
Snap/Grid	
Grid mode	<input type="checkbox"/> Grid on
Grid unit	0.5, 0.5
Grid major	5
Grid display	0x0003 (3)
Grid style	0x0000 (0)
Grid axis color	251
Grid major color	252
Grid minor color	253
Grid XYZ tint	0x0001 (1)
Snap angle	0.0000
Snap base	0.0, 0.0
Snap isometric pair	[0] Left
Snap mode	<input type="checkbox"/> Snap on (for current viewport)
Snap style	[1] Isometric snap
Snap type	[0] Grid snap
Snap unit	0.5, 0.5
Entity snap coordinates	[2] Keyboard entry overrides entity snap settings except in sc
Ignore entity snap elevation	<input type="checkbox"/> Replace Z value with current elevation
Polar distance	0.0

2. Turn on the *Orthogonal Mode*.
3. Use the *SNAP* command to set the *Snap Style* setting to *Isometric Snap*.

4. Press the *F5* function key to set the appropriate drawing plane: *Top*, *Left* or *Right*.



Draw isometric: Learn how to draw an isometric perspective using Polar Tracking and Entity Snap Tracking.

Setting the Snap Angle

You can rotate a dotted grid using the Snap Angle setting (if *Snap Angle* is different from zero, grid lines do not display).

5. Display the *Snap/Grid* settings in the Settings dialog:
6. Select the *Snap Angle* setting.
7. Type a new value in the *Snap Angle* setting field.
8. Close the *Settings* dialog.

NOTE You can also set the *Snap Angle* by typing *snapang* in the command bar.
When in a command, type '*snapang*' (with an apostrophe in front), to set the *Snap Angle* variable transparently (= without interrupting the running command).

Entity Snaps

Commands: 2DINTERSECTION, 3DINTERSECTION, NEAREST, CENTER, ENDPOINT, EXTENSION, FROM, INSERTION, INTERSECTION, MIDPOINT, MTP, NONE, PARALLEL, PERPENDICULAR, POINT, QUADRANT, TANGENT

Entity snaps enable you to quickly select exact geometric points on existing entities without having to know the exact coordinates of those points. With entity snaps, you can select the end point of a line or arc, the center point of a circle, the intersection of any two entities, or any other geometrically significant position. You can also use entity snaps to draw entities that are tangent or perpendicular to an existing entity. You can use entity snaps any time you need to specify a point.

You can work with entity snaps in one of two ways

- Enable a running entity snap that remains in effect until you turn it off by choosing an entity snap when no other command is active.
- Enable a one-time entity snap for a single selection by choosing an entity snap when another command is active. You can also use a one-time entity snap to override a running entity snap.

When using entity snaps, the program recognizes only visible entities or visible portions of entities. You cannot snap to entities on layers that have been turned off or to the blank portions of dashed lines.

If the *APBOX* system variable is on, the entity *Snap Aperture Box* is added to the crosshairs when one or more entity snaps are active.

When you move the cross hairs, the program snaps to the snap point closest to the center of the *Snap Aperture Box*. The *Snap Marker* indicates the current snap point.

The DRAGSNAP system variable controls the snap behavior while 'dragging', providing an increased WYSIWIG experience. DRAGSNAP controls whether rubberband dynamics are displayed at the current cursor location or at the current entity snap location. DRAGSNAP applies to all modify commands that display dynamics, such as COPY, PASTECLIP, PASTEBLOCK, MOVE, ROTATE, MIRROR, SCALE and STRETCH.

- DRAGSNAP = 0 (Default): dragged entities display at the cursor location.
- DRAGSNAP = 1: dragged entities display at the current entity snap location.

Hatch pattern snap: snapping to hatch pattern geometry is controlled by the OSOPTIONS system variable. The default value of OSOPTIONS is 3.

- OSOPTIONS = 0: Object snapping is enabled for hatch entities, and geometry with negative Z values when using a dynamic UCS.
- OSOPTIONS = 1: Object snapping is disabled for hatch entities.
- OSOPTIONS = 2: Object snapping is disabled for geometry with negative Z values when using a dynamic UCS.
- OSOPTIONS = 3: Object snapping is disabled for hatch entities, and geometry with negative Z values when using a dynamic UCS.

NOTE Press the *TAB* key, without moving the mouse, to cycle through all possible entity snaps. If the *DYNMODE* variable has a positive value (Dynamic Dimensions switched ON) entity snap cycling is not possible. Click the *DYNMODE* field in the Status Bar to toggle the display of dynamic dimensions.

To define the Entity Snap settings

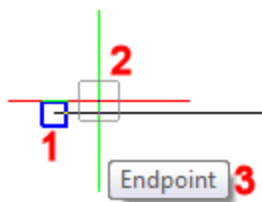
1. Open the Settings dialog.
2. In the *Settings* dialog, expand the *Program Options* settings class.
3. Under *Program Options* expand the *Display* settings group, then scroll down to the *Entity Snap* settings.

Snap marker size	6
Snap marker thickness	2
Snap marker color	■ 20
Auto tracking vector color	■ 171
Snap marker in all views	<input type="checkbox"/> Display snap marker in all views
Snap tooltips	<input checked="" type="checkbox"/> Enable snap tooltips

Under *Drawing* expand the *Drafting* settings group then go to *Coordinate Input > Entity Snaps*.

Entity snaps	
Entity snap mode	0x0003 (3)
Entity snap aperture box	<input type="checkbox"/> Display Aperture box
Entity snap aperture	10
Pdf entity snap	<input checked="" type="checkbox"/> Enable PDF entity snap

4. Define the *Entity Snap* settings.

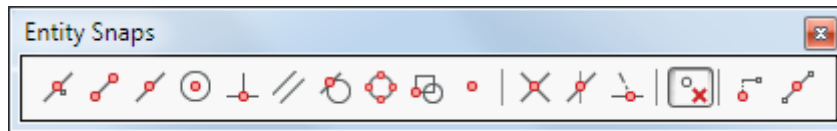


User Preference	Title	Description
SnapMarkerSize	Snap marker size	Sets the size of the snap marker. (default size is 6)
SnapMarkerThickness	Snap marker thickness	Sets the thickness of the snap marker. (default thickness is 2)
SnapMarkerColor	Snap marker color	Sets the color of the snap marker. (default color = 20)
DisplaySnapMarkerInAllViews	Snap marker in all views	If multiple viewports are open, enables the display of the snap marker in all viewports.
DisplayTooltips	Snap tooltips (3)	Enables the display of the Entity Snap tooltips.
System Variable	Title	Description
OSMODE	Entity snap mode	Sets the entity snap modes and toggles entity snap on/off.
APBOX	Entity snap aperture box	Toggles the display of the aperture box (2).
APERTURE	Entity snap aperture	Sets the size of the Autosnap aperture box. (default size is 10)
OSNAPZ	Ignore entity snap elevation	Replaces the Z-coordinate of the snapping point with the current value of the ELEVATION system variable.
PDFOSNAP	Pdf entity snap	Enables snapping to the geometry in pdf underlay files.

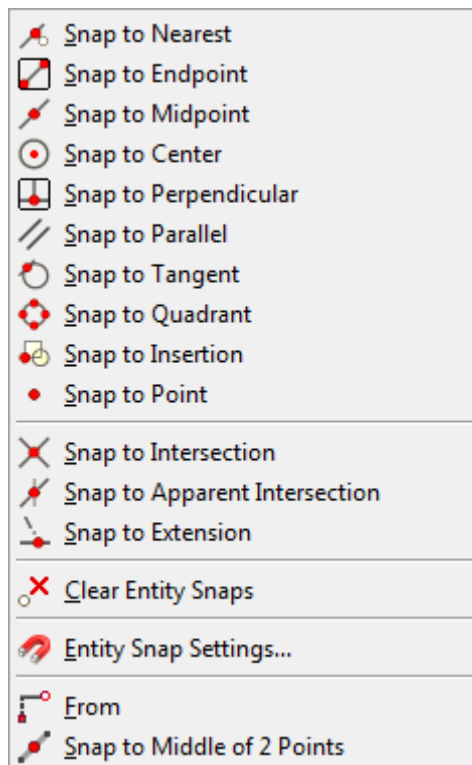
To set the Entity Snaps

Do one of the following:

- Click the buttons on the *Entity Snaps* toolbar.
The buttons of the currently active entity snap modes are pressed.



- Press and hold the Shift key, then right click and select the Entity Snap mode in the context menu.
The icons of the currently active Entity Snap modes are outlined.



When no command is active, the above procedures toggle the Entity Snap modes on/off.

When a command is active, the above procedures set a 'one shot' override of the current Entity Snap modes.

Alternatively you can type a single letter shortcut as indicated in the table below.

Entity Snap Modes

Name	Icon	Marker	Context menu shortcut	Description
NEAREST			N	Snaps to the nearest point on an entity.
ENDPOINT			E	Snaps to the nearest endpoint of an entity or polyline segment.
MIDPOINT			M	Snaps to the midpoint of an entity or polyline segment.
CENTER			C	Snaps to the center point of an arc, circle, polygon, ellipse or elliptical arc. Snaps to the center of gravity of a closed polyline.
PERPENDICULAR			P	Snaps to the perpendicular point of another entity. You can snap to an arc, circle, ellipse, line, polyline, infinite line, ray, spline or edge of a plane to form a perpendicular alignment with that entity or with an extension of that entity.
PARALLEL			L	Displays a tracking line parallel to the selected entity.
TANGENT			T	Snaps to the point on an arc, ellipse, spline or circle that, when connected to the previous point, forms a line tangent to that entity.
QUADRANT			Q	Snaps to the closest quadrant of an arc, circle, ellipse, or elliptical arc.
INSERTION			I	Snaps to the insertion point of an attribute, block or text entity.
POINT			O	Snaps to a point entity.
INTERSECTION			R	Snaps to the intersection of any combination of entities.
APPARENT INTERSECTION			A	Snaps to the apparent intersection in the current view of two entities that do not intersect in three-dimensional space.
EXTENSION			X	Snaps to the extension of an entity or to the intersection of the extension of two entities.
NONE			Y	Turns off all entity snap modes.
FROM			F	Prompts for a base point. Use relative coordinates to define the offset from the base point.
MTP			2	Snaps to a point in the middle between two points.

NOTE You can toggle the *Entity Snaps* on/off:

- by clicking the *ESNAP* field in the Status Bar.
- by pressing the *F3* function key.

Working with multiple Entity Snap modes

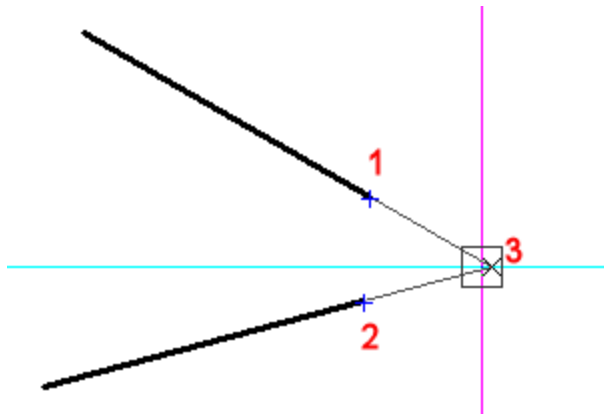
1. Move the cursor to the entity you want to snap.
One of the active Entity Snap modes markers display.
2. Do not move the mouse; press the TAB key.
The entity the snap point is on highlights.
3. Do one of the following:
 - Click to accept the snap point.
 - Press the TAB key.
The next possible Entity Snap mode marker displays.
The entity the snap point is on highlights.
4. (option) Repeat step 3 until the right snap point is found.

NOTES

- Do not move the mouse while pressing the TAB key.
- Entity snap cycling is not possible when Dynamic Dimensions are active. Click the *DYNMODE* field in the Status Bar to toggle Dynamic Dimensions on/off.

To snap to the extension of two entities

1. If not already on, turn on the *Extension* Entity Snap mode (↔-).
2. Launch a drawing tool, e.g. Draw Line.
3. Move the cursor over the endpoint of the first entity (1).
A small cross (⊕) indicates the entity is marked for extension.
4. Move the cursor over the endpoint of the second entity (2).
A small cross (⊕) indicates the entity is marked for extension.
5. Move the cursor near the intersection of the extensions of the two entities.
An X indicates the intersection (3).




6. Click to accept the snap point.

NOTE You can snap to the extension of lines, polylines, arcs and elliptical arcs

Using the From option

The *From* entity snap option allows to define an offset from an entity snap point.



1. When executing a drawing or modifying command, do one of the following:
 - Click the *From* tool () button on the *Entity Snaps* toolbar.
 - Type *from* in the command bar, then press Enter.
 - Press Shift, then right click and choose *From* in the context menu.

The command bar reads: Base point:


2. Identify the base point.
The command bar reads: Base point: offset or regular point:
3. Type the offset, from the base point as relative coordinates: @x,y,(z).

NOTE Use polar tracking to type the distance from the base point (see: Direct distance entry).

Using the Parallel entity snap

1. Make sure Entity Snap Tracking is active.
2. Launch an entity create command to create a linear entity (e.g. Line).
3. Specify the first point of the entity.
The line displays dynamically.
4. Do one of the following:
 - Press and hold the Shift key, then right click and choose *Snap to parallel* in the context menu.
 - Click the *Snap to parallel* toolbutton () in the *Entity Snaps* toolbar.
5. Hover the cursor over the entity you want to draw parallel to.
The *Parallel Entity Snap* marker () displays on the selected entity.
Wait until a small cross (+) indicates the entity is recognized.
6. Move the cursor in order to place the dynamics of the line more or less parallel to the selected entity.
A tracking line displays from the start point of the line.
7. Do one of the following to specify the second point while the tracking line displays:
 - Click a point.
 - Enter the desired length and press Enter.
 - Use a second entity snap tracking point, then move the cursor to the intersection of the parallel tracking line and the second entity snap tracking line.

Using Midpoint snap

1. When prompted to specify a point, do one of the following:
 - Click the *Snap to the middle of 2 points* button () on the *Entity Snaps* toolbar.
 - Type *mtp* or *m2p* in the command bar, then press Enter.
 - Press and hold the Shift key, then right click and choose *Snap to middle of 2 points* in the context menu.

The command bar reads: First point of mid:
2. Specify the first point
The command bar reads: Second point of mid:
3. Specify the second point.
The point in the middle of the two points is snapped.

Polar Tracking

Polar tracking assists you to draw at exact angles:

- at fixed intervals, starting from 3 O'clock (or East)
- at specific additional angles

A polar tracking path line displays from the origin point when you move the cursor close to one of the polar tracking angles. The tracking path lines display as long as the *Snap Aperture* box overlaps the polar tracking line.

In 3D views an additional tracking line displays parallel to the Z-axis of the current UCS.

Snap track lock: to lock a snap track path press the Shift key. To unlock a track path, press Shift again: it works as a toggle. Locked tracking paths turn red.

NOTE The display of the tracking path line is controlled through the *TRACKPATH* settings variable.
The color of the tracking path line is controlled through the *AutoTrackingVecColor* user preference.



Draw isometric (Internet connection needed)

Learn how to draw an isometric perspective using Polar Tracking and Entity Snap Tracking.

To define the Polar Tracking settings

- Do one of the following:
 - Right click on the *POLAR* field in the Status Bar, then right click and choose *Settings* in the context menu.
 - Open the *Settings* dialog, then expand the *Coordinate input* sub-category under *Drafting* in the *Drawing* settings class. Expand the *Snap Tracking* settings group.

Snap tracking	
AutoSnap	0x003F (63)
1	<input checked="" type="checkbox"/> AutoSnap marker
2	<input checked="" type="checkbox"/> AutoSnap tooltips
4	<input checked="" type="checkbox"/> AutoSnap magnet
8	<input checked="" type="checkbox"/> Polar tracking
16	<input checked="" type="checkbox"/> Entity snap tracking
32	<input checked="" type="checkbox"/> Tooltips for polar tracking and entity snap tracking
Polar angle	45.0
Polar add angles	30;60
Polar mode	0x000F (15)
1	<input checked="" type="checkbox"/> Relative
2	<input checked="" type="checkbox"/> Use polar tracking settings in entity snap tracking
4	<input checked="" type="checkbox"/> Use additional polar tracking angles
8	<input checked="" type="checkbox"/> Press SHIFT to acquire entity snap tracking points
Track path	[0] Display full-screen entity snap tracking path

- Adjust the settings:
 - 1: Autosnap marker (not used, required for AutoCad® compatibility only)
 - 2: Autosnap tooltips (not used, required for AutoCad® compatibility only)
 - 4: Autosnap magnet (not used, required for AutoCad® compatibility only)
 - 8: If checked, *Polar Tracking* is on.

- 16: If checked, *Object Snap Tracking* is on.
- 32: If checked, tooltips display for polar tracking and entity snap tracking.
- Polar angle: Sets the polar angle increment.
- Polar add angles: Sets the additional polar angles. Separate angles by semicolons.
- 1: If checked, polar angles are measured relatively from selected entities.
- 2: If checked, the polar angles (interval and additional angles) are also used in entity snap tracking.
- 4: If checked, the additional angles for polar tracking are used.
- 8: If checked, press and hold the Shift key to acquire entity tracking points.
- Track path: Controls the display of the polar and entity snap tracking paths.

Display full-screen object snap tracking path Display object snap tracking path only between the alignment point and the From point to the cursor location Do not display polar tracking path Do not display polar or object snap tracking paths

Using Polar Tracking to draw a line

1. (option) Check the *Polar Tracking* settings.
2. (option) If not already on, click the *POLAR* field in the Status Bar.
3. Launch the *Draw Line* tool.
4. Specify the start point of the line.
Polar tracking lines display at the specified intervals and additional angles.

If the cursor is on or next to a tracking line:

- The current distance and angle from the start point display next to the cursor when
 - A cross marker (X) indicates the current point.
5. Do one of the following to define the endpoint of the line.
When the appropriate tracking line displays:
 - Click when the cursor is at the desired position.
 - Type the length of the line in the command bar and press Enter.

NOTE Turning on *Polar Tracking* automatically disables Orthogonal Mode.

Using Snap Track Lock

Holding down the Shift key pressed while a snap track marker line is displayed, will keep this snap tracker locked, any (snap)point that is picked will be projected perpendicularly onto the tracking line.

A locked tracking line changes to the color defined by the SnapMarkerColor user preference.

Using temporary tracking points

When *Polar Tracking* is ON, temporary tracking points can be used to pick points at locations where no existing geometry is available to snap directly to, while you know the desired offset distance(s) from snap points on existing geometry in the neighborhood.

TT option: whenever prompted to enter a point, type *TT* on the command line, then press Enter to define a single temporary tracking point. After the tracking point has been picked, normal command execution continues. Repeat typing *TT* to enter additional temporary tracking points.

TK option: whenever prompted to enter a point, type *TK* on the command line, then press Enter to define one or more temporary tracking points. When eventually the desired location has been reached, enter the point by pressing the Right Mouse button or the Enter key.

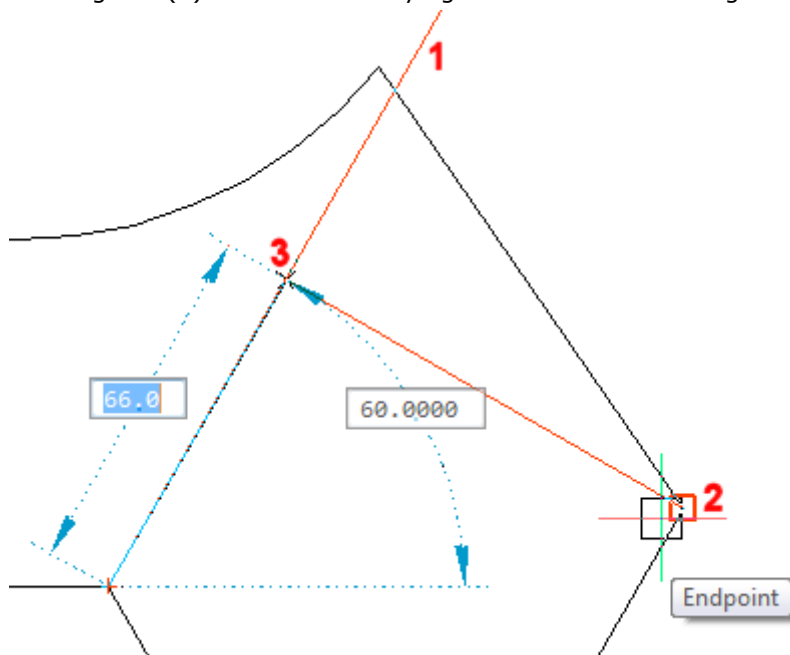
Using the TT option

1. Launch a draw command, e.g. Line.
The command bar reads: ENTER to use last point/Follow/<Start of line>:
2. Type *TT* in the command bar, then right click or press Enter.
The command bar reads: Specify temporary tracking point:
3. Use an Entity Snap to pick a point on existing geometry.
A small red cross indicates the temporary tracking point.
4. Move the cursor in the desired direction.
Depending on the value of the POLARANG and POLARADDANG system variables tracking lines display from the first temporary tracking point.
5. When a tracking line in the appropriate direction displays, type the distance from the temporary tracking point.
6. Specify the endpoint of the line.

Using the TK option

1. Launch a draw command, e.g. Line.
The command bar reads: ENTER to use last point/Follow/<Start of line>:
2. Type *TK* in the command bar, then right click or press Enter.
The command bar reads: Specify first temporary tracking point:
3. Use an Entity Snap to pick a point on existing geometry.
A small red cross indicates the temporary tracking point.
The command bar reads: Specify next temporary tracking point (press ENTER to end):
4. Move the cursor in the desired direction.
Depending on the value of the POLARANG and POLARADDANG system variables tracking lines display from the first temporary tracking point.
5. When a tracking line in the appropriate direction displays, type the distance from the temporary tracking point.
A small red cross indicates the second temporary tracking point.
The command bar reads: Specify next temporary tracking point (press ENTER to end):
6. Do one of the following:
 - Right click or press Enter to accept the second tracking point as the start point of the line.
 - Repeat steps 4 and 5 until the desired point is specified, then right click or press Enter to accept the point.
7. Specify the endpoint of the line.

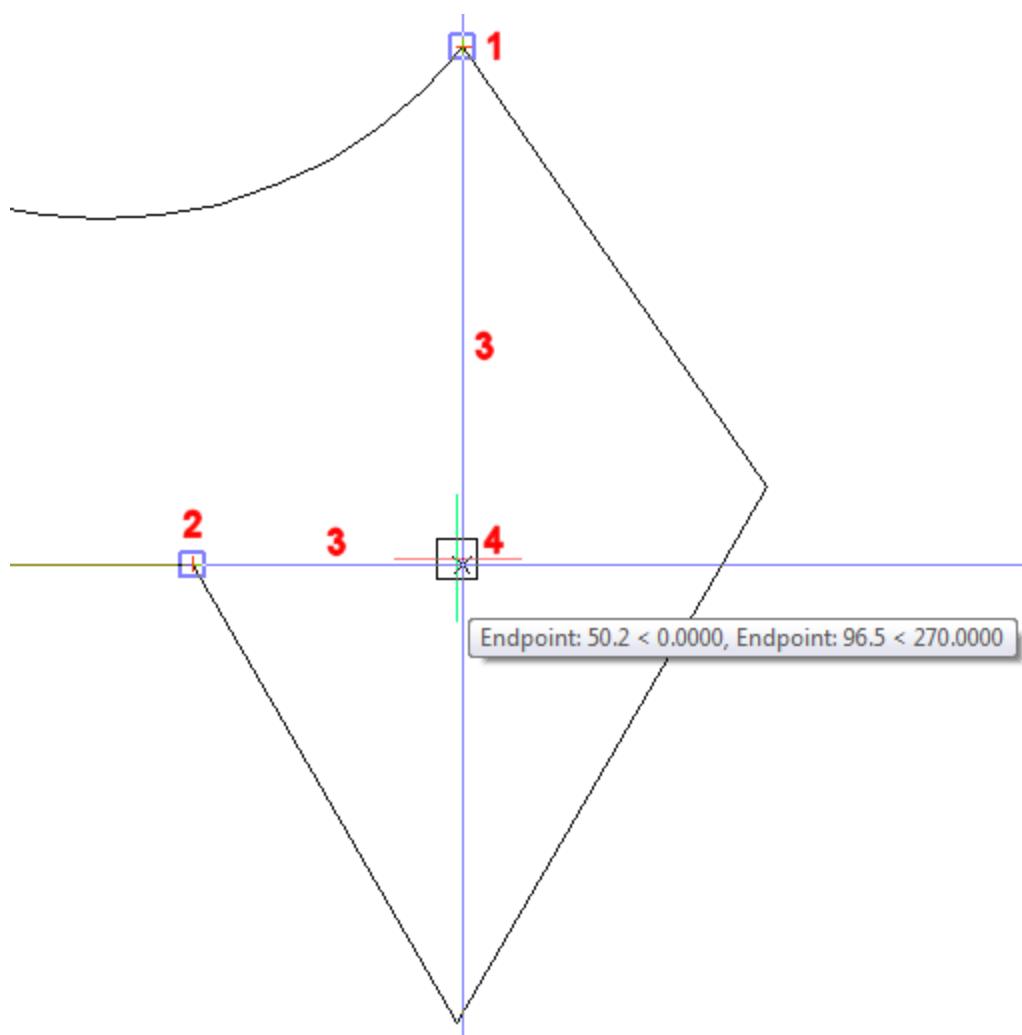
Hitting the Shift key when a tracking line displays, locks the tracking line. Lock tracking lines display in red (1). Entity snaps (2) or specified points are projected perpendicularly onto the tracking line (3). Hit the Shift key again to unlock a tracking line.



Using Entity Snap Tracking

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- An *Entity Snap Tracking* tooltip at the intersection of the tracking lines shows the distances from the tracking points.



5. Click to accept the *Entity Snap Tracking* point.

User Coordinate Systems

When you create entities in a drawing, they are located in relation to the drawing's underlying Cartesian coordinate system. The *ELEVATION* system variable specifies the distance from the xy-plane of the current coordinate system. If *ELEVATION* = 0 (zero), entities are created in the xy-plane of the current coordinate system.

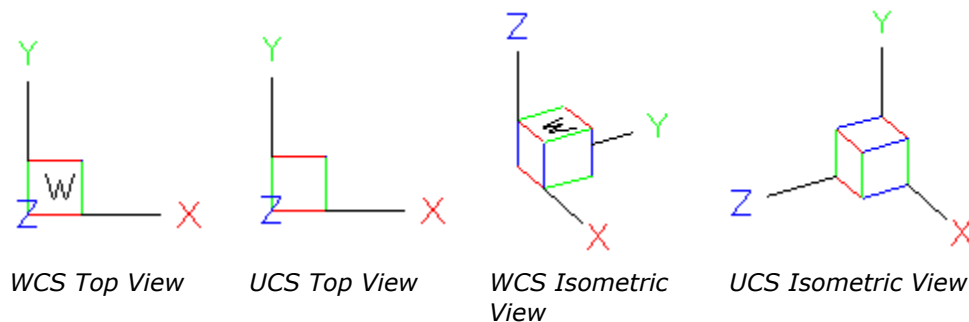
Every drawing has a fixed coordinate system called the World Coordinate System (WCS).

You can also define arbitrary coordinate systems located anywhere in three-dimensional space. These are called user coordinate systems (UCS) and can be located anywhere in the WCS and oriented in any direction. You can create as many UCS as you want, saving or redefining them to help you construct three-dimensional entities. By defining a UCS within the WCS, you can simplify the creation of most three-dimensional entities into combinations of two-dimensional entities.

When you create a new drawing the WCS is the current coordinate system by default, which is indicated by the letter W in the UCS icon. When you display a drawing in plan view, you see the coordinate system icon from the top, with the z-axis directed straight toward you. When you display a three-dimensional drawing in a view other than plan view, the coordinate system icon changes to reflect your new viewpoint.

NOTE You cannot delete or modify the WCS.

To help you keep your bearings in the current coordinate system, BricsCAD displays a coordinate system icon (UCS icon). The visible portions of the axes are the positive directions.



Three colors represent the three axes, making it easier for you to recognize the orientation in three-dimensional space:

- x-axis: red
- y-axis: green
- z-axis: blue

The Plan View command restores the Plan view or Top view of the current UCS or WCS.

The *UCSICON* settings variable controls the display and location of the UCS icon:

UCS icon	0x0003 (3)
0x0001	<input checked="" type="checkbox"/> Show icon
0x0002	<input checked="" type="checkbox"/> at origin
UCS icon position	Lower right

Show icon:

Controls whether the icon shows or not.

at origin:

Controls the location of the UCS icon: if on, the icon indicates the origin point of the current coordinate system (UCS or WCS). However, if the origin is not within the viewport borders, the UCS icon moves to the corner of the viewport, as defined by the *UCSICONPOS* (UCS Icon Position) settings variable.

When the *at origin* option is not checked, the icon always displays in the corner of the viewport defined by *UCSICONPOS*.

Dynamic UCS

With the Dynamic UCS feature enabled, the coordinate system is aligned automatically with a face of a solid when drawing entities.

This feature is controlled by the UCSDETECT system variable.

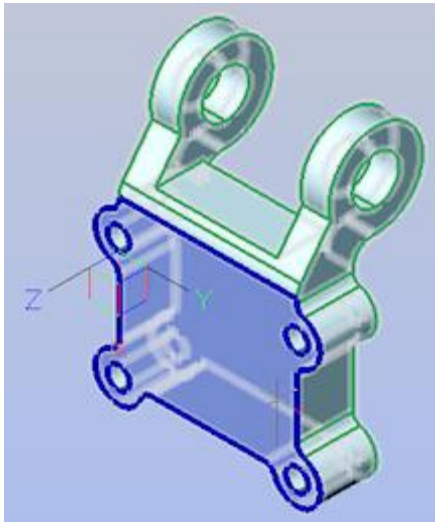
To toggle the dynamic UCS

Do one of the following:

- Press the *F6* function key.
- Click the *DUCS* field in the status bar.

Using the Dynamic UCS

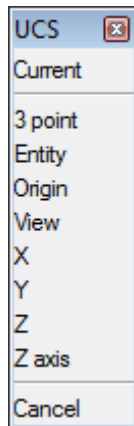
1. With the Dynamic UCS feature active, launch a Draw command.
2. Move the cursor over the face of a 3D solid.



- The face highlights.
 - If ON, the grid is aligned with the selected face.
 - If the UCSICON system variable = 3 (On at Origin), the UCS icon displays in the selected face.
3. Pick a point to start drawing.
 4. When the drawing command concludes, the current UCS or WCS is restored.

To define a User Coordinate System

1. Choose *Drawing Explorer* > *Coordinate Systems* in the *Tools* menu.
The *Drawing Explorer - Coordinate Systems* dialog opens.
2. Click the *New* tool button () on the *Drawing Explorer* dialog.
The *BricsCAD Explorer* dialog closes.
The command bar reads: Current/Entity/Origin/View/X/Y/Z/ZAxis<3point>:
A prompt menu displays:



3. Press Enter to define the UCS using points.
The command bar reads: New origin <current origin>:
4. Specify the origin point of the UCS.
The command bar reads: Point of positive X axis <current point>:
5. Specify a point to define the positive X-axis.
The command window reads: Point in X-Y plane with positive Y value <current point>:
6. Specify a point to define the positive Y-axis.
The UCS is defined.
The *BricsCAD Explorer* dialog reopens.
7. Click in the *UCS Name* field of the newly defined UCS to replace the <NewUCS> default name.
8. Close the *BricsCAD Explorer* dialog.

Command Options


See the EXPUCS and UCS commands.

To restore the WCS

1. Type *UCS* in the command bar, then press Enter.
The command bar reads:
?/3point/Delete/Entity/Origin/Previous/Restore/Save/View/X/Y/Z/Zaxis/<World>:
A prompt menu displays.
2. Do one of the following:
 - Press Enter to accept the default option.
 - Choose *World* in the prompt menu.
 - Type *W* in the command bar and press Enter.

To restore a UCS

1. Choose *Drawing Explorer > Coordinate Systems* in the *Tools* menu.
The *Drawing Explorer - Coordinate Systems* dialog opens.
2. Click the blank tile in front of the UCS name to make it current.

Details				
				
	UCS Name	Origin (WCS)	X Axis Direction (WCS)	Y Axis Direction (WCS)
1	Back	0.0000, 0.0000, 0.0000	-1.0000, 0.0000, 0.0000	0.0000, 0.0000, 1.0000
2	Front	0.0000, 10.0000, 0.0000	1.0000, 0.0000, 0.0000	0.0000, 0.0000, 1.0000
3	Left	10.0000, 0.0000, 0.0000	0.0000, -1.0000, 0.0000	0.0000, 0.0000, 1.0000
4	✓ Right	0.0000, 0.0000, 0.0000	0.0000, 1.0000, 0.0000	0.0000, 0.0000, 1.0000
5	Rotated	0.0000, 0.0000, 10.0000	-0.7071, 0.7071, 0.0000	0.4082, 0.4082, 0.8165

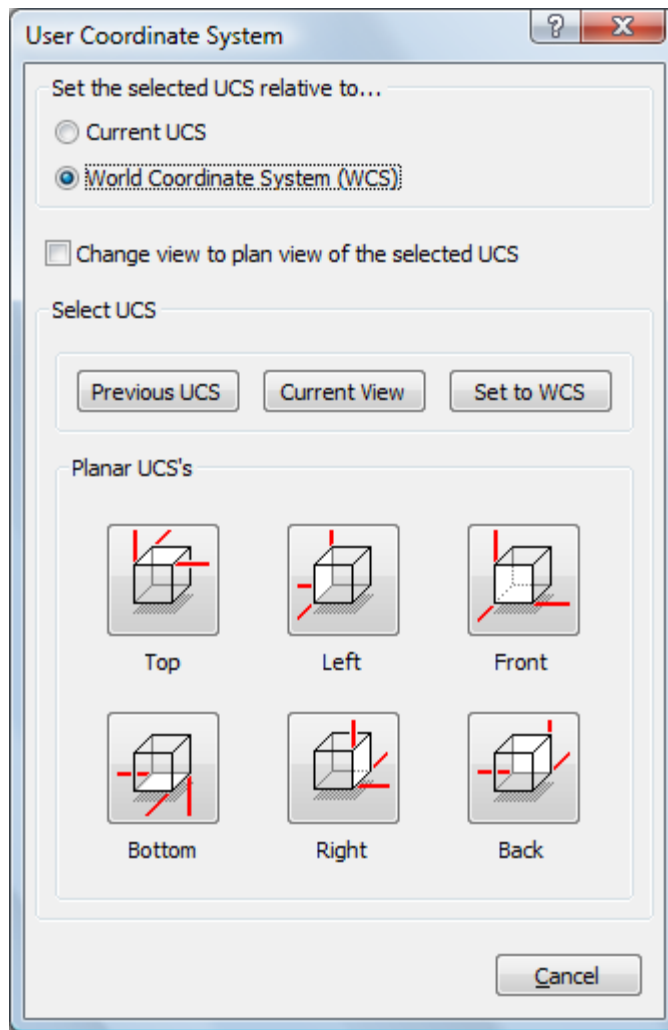
3. Close the *BricsCAD Explorer* dialog.
- 4.

NOTE If no UCS is marked, the WCS is the current coordinate system.

To set a relative UCS

1. Do one of the following:
 - Click the *UCS...* tool button () on the *Settings* toolbar.
 - Choose *UCS...* in the *Settings* menu.

The *User Coordinate System* dialog opens.



2. Choose one of the following:
 - Set the selected UCS relative to the *Current UCS*.
 - Set the selected UCS relative to the *World coordinate system (WCS)*.
3. (option) Enable the *Change view to plan view of the selected UCS* option.
4. Click one of the *Planar UCS's* buttons.
The User Coordinate System dialog closes.

- NOTES**
- If the UCSORTHO system variable is ON an orthographic UCS is restored automatically when an orthographic view is restored by the -VIEW command or by one of the tools on the Look From toolbar.
 - The UCSVP (UCS viewports) system variable controls whether the UCS in a viewport is fixed (UCSVP = ON) or changes to match the UCS in the current viewport (UCSVP = Off). As a result, in a multiple viewport drawing, the UCS of all viewports that have UCSVP = Off matches the orthographic UCS of the current viewport if UCSORTHO = On.

Command Options

See the SETUCS command.

Measuring

Commands: DIST, AREA, MASSPROP, ID and LIST

The *Dist* command reports the distance and angle between two points.

The *Area* command finds the area and perimeter (or length) of 2D entities.


The *Massprop* command reports the area, perimeter, and other mathematical properties of 3D solids and 2D regions (short for "mass properties").

The *ID* command reports the x,y,z coordinates of picked points.

The *List* command lists the properties of selected entities.

Measuring distances

To measure a distance between two points:

1. Do one of the following:
 - Click the *Distance* tool button () on the *Inquiry* toolbar.
 - Type *dist* in the command bar and press Enter.

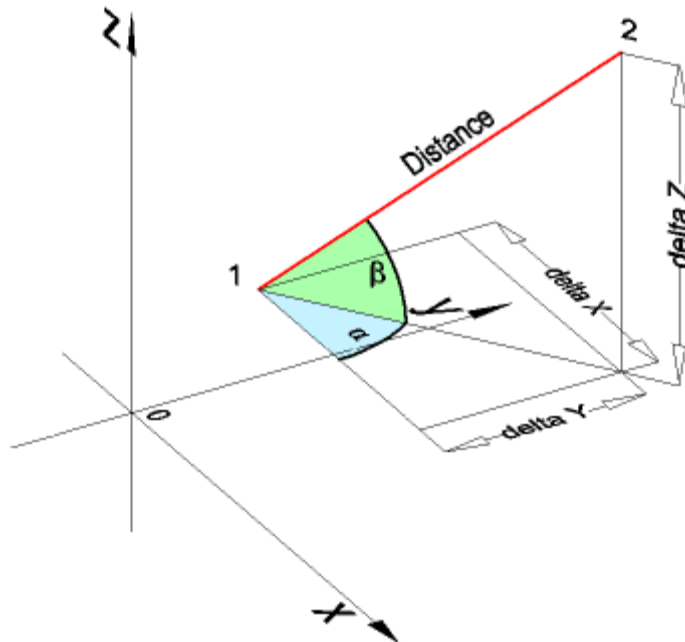
The command bar reads: Starting point for distance:

2. Identify the first point.
The command bar reads: End point:
3. Identify the second point.

The following information displays in the command bar (see image below):

Distance, Angle in XY Plane (alpha), Angle from XY Plane (beta),
Delta X, Delta Y, Delta Z.


The values are calculated with respect to the current coordinate system.



Measuring lengths

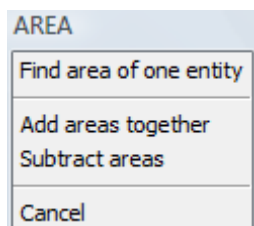
To measure the length of single linear entity, click the entity then read the *Length* field in the Properties Bar.

To add the lengths of a series of linear entities:

1. Do one of the following:
 - Click the *Area* tool button () on the *Inquiry* toolbar.
 - Type *area* in the command bar and press Enter.

The command bar reads: Entity/Add/Subtract/<First point>:

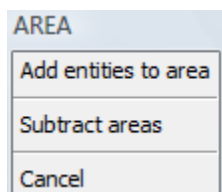
A prompt menu displays:



2. Do one of the following:
 - Choose *Add areas together* in the prompt menu.
 - Type *A* in the command bar and press Enter.

The command bar reads: Adding: Entity/Subtract/<First point>:

A prompt menu displays:




3. Do one of the following:
 - Choose *Add entities to area* in the prompt menu.
 - Type *E* in the command bar and press Enter.

The command bar reads: Adding area<Select entities>:
4. Click the first entity.
The area and length of the first entity display in the command bar.
The command bar reads: Adding area<Select entities>:
5. Click the second entity.
The area and length of the second entity and the total area and total length display in the command bar.
Optionally press the *F2* function key to display the *Prompt History* window; press *F2* again to close the *Prompt History* window.
The command bar reads: Adding area<Select entities>:
6. Repeat step 5 to add more entities
or
press the *Esc* key to stop.

Measuring areas

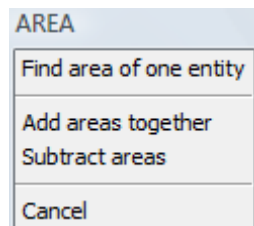
To measure the area and length of single closed entity, click the entity then read the *Area* and *Length* fields in the Properties Bar.

To measure areas and perimeters by specifying points in a drawing:

1. Do one of the following:
 - Click the *Area* tool button () on the *Inquiry* toolbar.
 - Type area in the command bar and press Enter.

The command bar reads: Entity/Add/Subtract/<First point>:

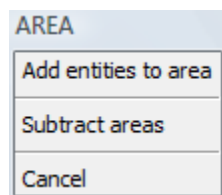
A prompt menu displays:



2. Do one of the following:
 - Choose *Add areas together* in the prompt menu.
 - Type *A* in the command bar and press Enter.

The command bar reads: Adding: Entity/Subtract/<First point>:

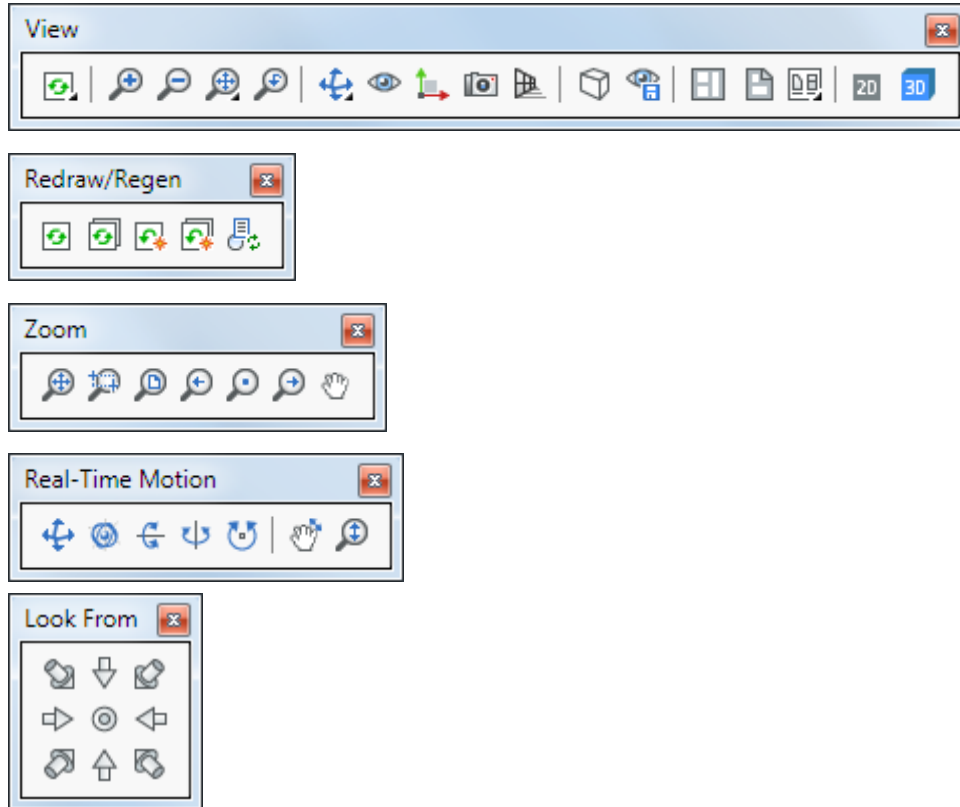
A prompt menu displays:



3. Click the first corner point of the area.
A dashed line displays dynamically from the first corner point.
The command bar reads: Adding: <Next point>:
4. Click the second corner point of the area.
A dashed polygon displays dynamically, indicating the currently defined area.
The command bar reads: Adding: <Next point>:
5. Click the third corner point of the area.
The command bar reads: Adding: <Next point>:
6. Repeat step 5 to add more corner points
or
Right click to stop adding corner points.
The *Total length* and *Total area* display in the command bar.
Optionally press the *F2* function key to display the *Prompt History* window; press *F2* again to close the *Prompt History* window.
7. Do one of the following:
 - Repeat steps 3 through 6 to add another area.
 - Press the *Esc* key to stop.

Viewing Your Drawing


















BricsCAD provides various tools to control the display and view orientation of your drawing. These tools can be found either in the *View* menu or the *View* toolbar or both.









View toolbar and its flyouts

Look From toolbar

Icon	Tool name	Command	Description
	Redraw	REDRAW	Refreshes the screen display in the current viewport.
	Redraw All	REDRAWALL	Refreshes the screen display in all open viewports.
	Regenerate	REGEN	Recalculates the screen display in the current viewport.
	Regenerate All	REGENALL	Recalculates the screen display in all open viewports.
	Update Fields	UPDATEFIELD	Updates the selected fields.
	Real Time Pan	RTPAN	Pans the view dynamically.
	Pan	PAN	Pans the view.
	Real Time Zoom	RTZOOM	Zooms in/out dynamically.
	Zoom In	ZOOM + I(*)	Zooms in on the center of the window by a factor of 2.









	Zoom Out	ZOOM + O(*)	Zooms out from the center of the window by a factor of 1/2.
	Zoom Extents	ZOOM + E(*)	Displays all the entities in the drawing (referred to as the <i>drawing extents</i>).
	Zoom Window	ZOOM + W(*)	You are prompted to pick two corners of a box on the existing view in order to enlarge that area to fill the display.
	Zoom Previous	ZOOM + P(*)	Restores the displayed view prior to the current one.
	Zoom All	ZOOM + A(*)	Displays the whole drawing as far as its drawing limits or drawing extents (whichever is the greater of the two).
	Zoom Left	ZOOM + L(*)	Pick the lower left corner of the next view, then specify the magnification factor or the view height.
	Zoom Center	ZOOM + C(*)	Pick the center of the next view, then specify the magnification factor or the view height.
	Zoom Right	ZOOM + R(*)	Pick the upper right corner of the next view, then specify the magnification factor or the view height.
	Look Around	RTLOOK	Look around in a 3D scene.
	Look Up and Down	RTUPDOWN	Move up / down and left / right in a 3D scene.
	Walk	RTWALK	Walk left / right and forward / backward in a 3D scene.
	Real-Time Constrained Sphere	RTROT	Rotates the view dynamically in 3D. <ul style="list-style-type: none"> Moving the mouse horizontally (parallel to the screen x-axis) rotates the view about the world z-axis. Moving the mouse vertically (parallel to the screen y-axis) rotates the screen x-axis.
	Real-Time Sphere Center	RTROTCTR	Rotates the view dynamically in 3D about a user-defined center point. Behaves like RTROT.
	Real-Time Free Sphere	RTROTFF	Rotates the view dynamically in 3D. <ul style="list-style-type: none"> Moving the mouse horizontally (parallel to the screen x-axis) rotates the view about the screen y-axis. Moving the mouse vertically (parallel to the screen y-axis) rotates the view about the screen x-axis.
	Real-Time X	RTROTX	Rotates the view dynamically about the screen x-axis.
	Real-Time Y	RTROTY	Rotates the view dynamically about the screen y-axis.
	Real-Time Z	RTROTZ	Rotates the view dynamically about the screen z-axis.
	Set View Point	DDVPOINT	Opens the <i>Preset Viewpoints</i> dialog.
	Plan View	PLAN	Restores the plan view with respect to the WCS, the current UCS or a saved UCS.
	Define View	DVIEW	Defines parallel and visual perspective views.
	Save/Restore view	VIEW	Saves and restores named views.

	Viewports	VPORTS	Manages viewports in model space.
	Paper Space Views	MVIEW	Creates viewports in paper space.
	Generate Drafting Views	BMGENDRAFT	Generates associative orthographic and standard isometric views of a 3D solid model in a paper space layout.
	Generate Section	BMGENSECTION	Creates a cross section view based on a standard drafting view generated by the <i>BmGenDraft</i> command in a paper space layout.
	2D Context	2DCONTEXT	Switches the drawing to a 2D drafting context.
	3D Context	3DCONTEXT	Switches the drawing to a 3D modeling context.

(*) <command> <option>: type *zoom* followed by Enter, then type the command option letter, followed by Enter.

Layer Utilities

The layer utility commands help you to easily manipulate the *On/Off*, *Thaw/Freeze* and *Unlocked/Locked* layer properties by selecting entities. You can then save the result in a Layer State.

Command	Icon	Description
LAYON		Turns on all layers that have been turned off
LAYOFF		Turns off layers associated with selected entities.
LAYFRZ		Freezes layers associated with selected entities.
LAYTHW		Thaws all frozen layers.
LAYLCK		Locks layers associated with selected entities.
LAYULK		Unlocks layers associated with selected entities.
LAYISO		Isolates layers associated with selected entities.
LAYUNISO		Restores layers isolated by the LAYISO command.



Using the Layer Tools (Internet connection needed)

Redrawing and Regenerating a Drawing

Commands: REGEN, REGENALL, REGENAUTO, REDRAW and REDRAWALL

The *Regen* command regenerates the current viewport (short for "regenerate").

The *Regenall* command regenerates all viewports (short for "regenerate all").

The *Regenauto* command controls when BricsCAD regenerates the drawing automatically.

The *Redraw* command redraws the current viewport to clean it up.

The *Redrawall* command redraws all viewports to clean them up.


As a matter of fact the screen display of a drawing is a simplified version of the drawing database. From time to time it is necessary to synchronise the screen display and the drawing database.

The *REGENMODE* system variable controls whether BricsCAD regenerates the drawing automatically. You can change the status of *REGENMODE* through the *RegenAuto* command. If *REGENMODE* is ON BricsCAD regenerates the display automatically, but in a few cases a forced regeneration of the drawing might be necessary. This is done by the *Regen* command.

Don't confuse the *Regen* command with the *Redraw* command, which simply repaints the screen, without attempting to synchronize the screen display with the drawing database.

Regenerate the current viewport


Do one of the following:

- Click the *Regen* tool button () on the *View* toolbar.
- Choose *Regen* in the *View* menu.
- Type *re* or *regen* in the command bar, then press Enter.

The current viewport is regenerated.

Regenerate all viewports


Do one of the following:

- Click the *Regenall* tool button () on the *View* toolbar.
- Type *rea* or *regenall* in the command bar, then press Enter.

All viewports are regenerated.

Redraw the current viewport


Do one of the following:

- Click the *Redraw* tool button () on the *View* toolbar.
- Choose *Redraw* in the *View* menu.
- Type *r* or *redraw* in the command bar, then press Enter.

The current viewport is refreshed.

Redraw all viewports


Do one of the following:

- Click the *Redraw All* tool button () on the *View* toolbar.
- Type *ra* or *redrawall* in the command bar, then press Enter.

All viewports are refreshed.

Panning

Commands: PAN and RTPAN


The *Pan* command moves the drawing in any direction: horizontally, vertically, or diagonally. The magnification of the drawing remains the same, as does its orientation in space. The only change is the portion of the drawing being displayed. The cursor changes to a hand () when a *Pan* tool is active.

The *Rtpan* command pans the drawing in real-time.

See also: View manipulation using the mouse.

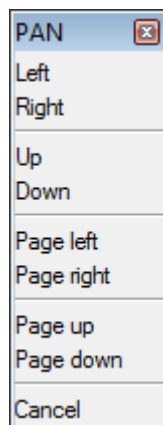
Using the pan command

1. To launch the Pan command do one of the following:

- Click the *Pan* tool button () on the *View* toolbar (*Zoom* flyout).
- Choose *Zoom > Pan* in the *View* menu.
- Type *p* or *pan* in the command bar, then press Enter.

The command bar reads: Left/Right/Up/Down/PGLLeft/PGRRight/PGUp/PGDown/<Pan base point>:


A prompt menu opens:



2. Specify the *Pan base point*.
The command bar reads: Pan displacement point:
3. Specify the *Pan displacement point*.
The display shifts over the specified distance and in the specified direction.

Using real time panning

1. To launch the Real-time Pan command do one of the following:

- Click the *Real-Time Pan* tool button () on the *Real-Time Motion* toolbar
- Choose *Real-Time Motion > Real-Time Pan* in the *View* menu.
- Type *rtpan* in the command bar, then press Enter.

The command bar reads: >>ENTER, Right click or Esc to complete...

2. Press and hold the left mouse button to pan the view.
3. To abort the *Real-Time Pan* command, do one of the following:
 - Right click.
 - On the keyboard, press Enter, space bar or Esc.


Zooming

Command: ZOOM and RTZOOM

The *Zoom* command visually changes the size of the drawing within the current viewport.

The *Rtzoom* command zooms the drawing in real-time.

You can change the magnification of your drawing at any time by zooming. The cursor

changes to a magnifying glass () when a *Zoom* tool is active. Zoom out to reduce the magnification so you can see more of the drawing, or zoom in to increase the magnification so you can see a portion of the drawing in greater detail. Changing the magnification of the drawing affects only the way the drawing is displayed; it has no effect on the dimensions of the entities in your drawing.

See also: View manipulation using the mouse.

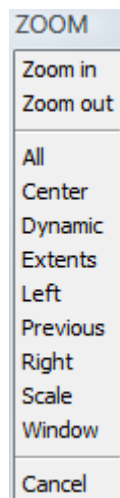
Using the zoom command

1. Do one of the following:
 - Choose *zoom* in the *View* menu.
 - Type *zoom* in the command bar.
 - Type *Z* in the command bar.

The command bar reads:


In/Out/All/Center/Dynamic/Extents/Left/Previous/Right/Window/<Scale (nX/nXP)>:

A prompt menu opens:



2. Do one of the following:
 - Type the zoom factor, followed by *X*. E.g. type *2x* to magnify the display 2 times; typing *0.5x* changes the display to half its original size.
 - In *paper space viewports*, type the zoom factor followed by *XP* to define the zoom factor relative to the viewport, thus defining the scale of the viewport content.
 - Pick two corners of a box on the existing view in order to enlarge that area to fill the display.

Real-time zooming

- Do one of the following:
 - Click the *Real-Time Zoom* tool button () in the View toolbar.
 - Type *rtzoom* in the command bar, then press Enter.

The command bar reads: >>ENTER, Right click or Esc to complete...
- Press and hold the left mouse button.
Move the mouse forward to zoom in.
Move the mouse backward to zoom out.
- To abort the *Real-Time Zoom* command, do one of the following:
 - Right click.
 - On the keyboard, press Enter, space bar or Esc.

Dynamic zooming

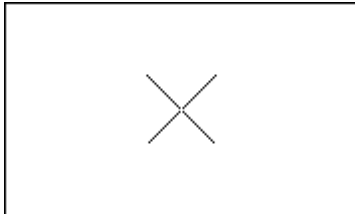
- Do one of the following:
 - Choose *zoom* in the *View* menu.
 - Type *zoom* in the command bar.
 - Type *Z* in the command bar.

A prompt menu displays.

The command bar reads:
In/Out/All/Center/Dynamic/Extents/Left/Previous/Right/Window/<Scale (nX/nXP)>:
- Do one of the following:
 - Choose *Dynamic* in the prompt menu.
 - Type *D*, then press Enter.

A view extents or view limits (if larger) is executed first.

A view box displays. The original size of the view box matches the size of the previous view.



A dashed rectangle indicates the area of the previous view.

- (option) To resize the view box:
 - Click to switch to resize mode. A right pointing arrow indicates you are in resize mode.



- Move the mouse to adjust the size.
 - Click to leave resize mode.
- Move the view box to the area you want to enlarge, then right click.
The drawing is zoomed in on the selected area.
 - Repeat steps 1 through 4 to zoom in on another part of the drawing.

View manipulation using the mouse and keyboard

In BricsCAD you can navigate through a 3D model using the mouse and keyboard shortcuts. In perspective mode you can walk inside the 3D model. In isometric mode you can only zoom in very close, but not go inside.

Mouse / Key	Action	Result
Wheel	scroll	Zoom in / out (*)
Middle Button or wheel (**)	press and hold while moving the mouse	Real-time pan (RTPAN command)
Middle Button or wheel (**)	double click	Zoom extents
Ctrl + Shift + Left Button	press and hold while moving the mouse	Real-time zoom in / out (RTZOOM command)
Ctrl + Shift + Right Button	press and hold while moving the mouse	Real-time pan (RTPAN command)
Ctrl + Shift + Middle Button or wheel	press and hold while moving the mouse	Real-time sphere (RTROT command)
Shift + Middle Button or wheel	press and hold while moving the mouse	Real-time constrained sphere (RTROT command)
Ctrl + Right Button	press and hold while moving the mouse	Real-time rotate about the screen Z-axis (RTROTZ command)
Walk through navigation		
Alt + Left Button	press and hold while moving the mouse	Walk left / right and forward / backward (RTWALK command)
Alt + Middle Button	press and hold while moving the mouse	Move up / down and left / right (RTUPDOWN command)
Ctrl + Middle Button	press and hold while moving the mouse	Look around (RTLOOK command)
Ctrl + Arrow Keys	press the arrow keys while holding the Ctrl key	Walk left / right and forward / backward (RTWALK command)
Ctrl + Shift + Arrow Keys	press the arrow keys while holding the Alt + Ctrl keys	Look up / down and left / right (RTUPDOWN command)
Ctrl + Home		Sets the view direction horizontal
Alt + Home		Moves target point to center of scene
Alt + plus/minus		Increase / decrease movement speed
Ctrl + plus/minus		Increase / decrease rotation speed

(*) The incremental change in zoom with each mouse-wheel action is controlled through the *Zoom Factor* variable (ZOOMFACTOR).

(**) On condition the *Middle Button Pan* (MBUTTONPAN) variable is set *ON*.

Setting the Zoom Factor variable

1. Type *zoomfactor* in the command bar, then press Enter.
The command bar reads: New current value for ZOOMFACTOR (3 to 100)
<current value>:

2. Type a value between 3 and 100 in the command bar, then press Enter.
or
3. In the Settings dialog, open the *Drawing* category, then expand the *Display/Viewing* sub-category.
In the *Viewing* group, select the *ZoomFactor* variable.
4. Type a value between 3 and 100 in the *zoomfactor* field.

NOTE Increase the zoom factor to speed up scroll wheel zooming in large drawings.

Setting the Middle Button Pan variable

1. Type in *mbuttonpan* the command bar, then press Enter.
The command bar reads: New current value for MBUTTONPAN (Off or On)
<current value>:
2. Type *On* or *Off* in the command bar.
or
3. In the Settings dialog, open the *Program Options* category, then expand the *User Preferences* sub-category and select the *MBUTTONPAN* variable.
4. Choose either *Support panning* or *Support action defined in menu file*.

View Rotation

Commands: RTROT, RTROTCTR, RTROTf, RTROTx, RTROTy, RTROTz, DDVPOINT, -VIEW, PLAN and VPOINT

The *Rtrot* (Real-Time Constrained Sphere) command rotates 3D drawings in real-time.

The *Rtrotctr* (Real-time Sphere Center) command freely rotates 3D drawings in real-time about a user-defined center point.

The *Rtrotf* (Real-time Free Sphere) command freely rotates 3D drawings in real-time.

The *Rtrotx*, *Rtrot y* and *Rtrotz* commands rotates 3D drawings in real-time about the x, y or z screen axis.

The *Ddvpoint* command sets 3D viewpoints or plan view, through a dialog box (short for "dynamic dialog view point").

The *Plan* command displays the plan viewpoint of drawings.

The *Vpoint* command changes the 3D viewpoint.

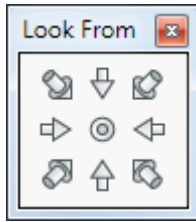
In order to view your 3D drawings from any angle, you can rotate a view. The *Real-Time Motion* tools of BricsCAD allow you to rotate a view in real-time. You can rotate the view about the X, Y or Z screen axis or in any direction (real-time sphere). If the *Continuous Motion* variable is set, the view rotation continues until you conclude the *Real-Time Motion* command.

See also: View manipulation using the mouse.

NOTE 'Real-time' commands should not be used when drawing in 2D. Use the Plan View tool to restore top view.



Restoring orthographic and isometric views

The tools on the *Look From* toolbar allow to restore orthographic and isometric views using the -VIEW command options.

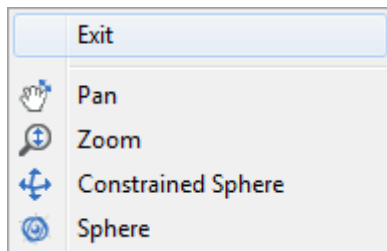


NOTE If the UCSORTHO system variable is ON the related orthographic UCS is restored automatically.

Rotating a view freely


1. Do one of the following:
 - Click the *Real-Time Constrained Sphere* tool button () or the *Real-Time Sphere* tool button () on the *Real-Time Motion* toolbar.
 - Choose *Real-Time Motion > Real-Time Constrained Sphere* or *Real-Time Motion > Real-Time Sphere* in the *View* menu.
 - Type *rtrot* , *rtrotf* or *rtrotctr* in the command bar, then press Enter.

The command bar reads: >> Press ENTER or Esc to complete, or right click to display context menu ...
2. Press and hold the left mouse button.
Move the mouse to rotate the view.
 - Using the *Real-Time Constrained Sphere* command horizontal mouse movement (parallel to the screen X-axis) rotates the 3D model about the world Z-axis.
 - Using the *Real-time Sphere Center* command you are prompted to specify a center point for the rotation first.
3. (option) Right click to display a context menu:

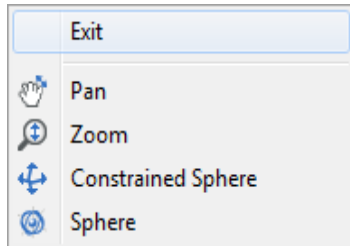


4. To abort the *Real-Time Sphere* commands, do one of the following:
 - Right click, then choose *Exit* in the context menu..
 - Press Enter, space bar or Esc.

Rotating a view about the view X-axis


1. Do one of the following:
 - Click the *Real-Time X* tool button () on the *Real-Time Motion* toolbar.
 - Choose *Real-Time Motion* > *Real-Time X* in the *View* menu.
 - Type *rtrotx* in the command bar, then press Enter.

The command bar reads: >> Press ENTER or Esc to complete, or right click to display context menu ...
2. Press and hold the left mouse button.
Move the mouse to rotate the view.
3. (option) Right click to display a context menu:

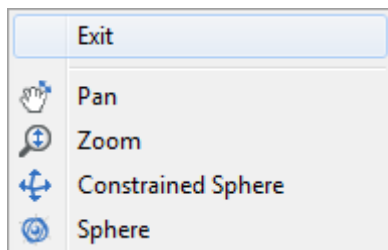


4. To abort the *Real-Time X* command, do one of the following:
 - Right click.
 - Press Enter, space bar or Esc.

Rotating a view about the view Y-axis


1. Do one of the following:
 - Click the *Real-Time Y* tool button () on the *Real-Time Motion* toolbar.
 - Choose *Real-Time Motion* > *Real-Time Y* in the *View* menu.
 - Type *rtroty* in the command bar, then press Enter.

The command bar reads: >> Press ENTER or Esc to complete, or right click to display context menu ...
2. Press and hold the left mouse button.
Move the mouse to rotate the view.
3. (option) Right click to display a context menu:

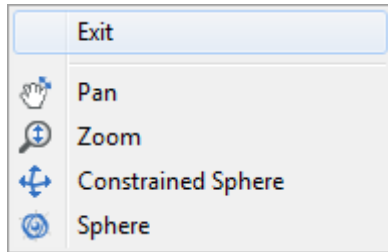


4. To abort the *Real-Time Y* command, do one of the following:
 - Right click.
 - Press Enter, space bar or Esc.

Rotating a view about the view Z-axis


- Do one of the following:
 - Click the *Real-Time Z* tool button () on the *Real-Time Motion* toolbar.
 - Choose *Real-Time Motion* > *Real-Time Z* in the *View* menu.
 - Type *rtrotz* in the command bar, then press Enter.


The command bar reads: >> Press ENTER or Esc to complete, or right click to display context menu ...
- Press and hold the left mouse button.
Move the mouse to rotate the view.
- (option) Right click to display a context menu:

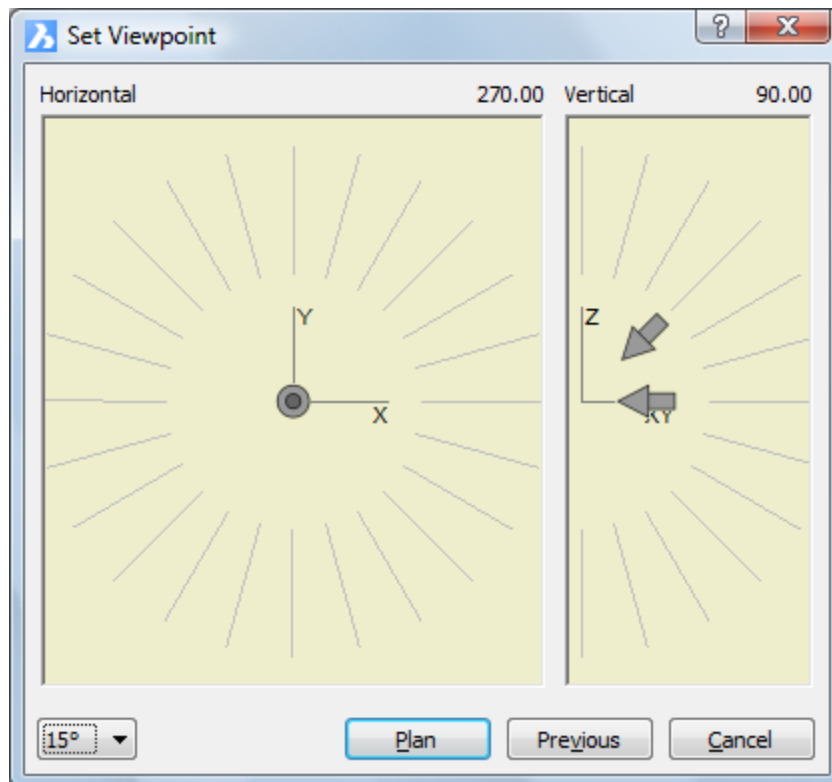


- To abort the *Real-Time Z* command, do one of the following:
 - Right click.
 - Press Enter, space bar or Esc.

Using Preset Viewpoints

- Do one of the following:
 - Click the *Set Viewpoint ...* tool button () on the *View* toolbar.
 - Choose *Set Viewpoint ...* in the *View* menu.
 - Type *ddvpoint* in the command bar, then press Enter.

The *Preset Viewpoint* dialog opens.
- Click the *Angle Precision* button () to choose a different display mode for the *Set Viewpoint* dialog.
Then choose either 45°, 15° or 5°.



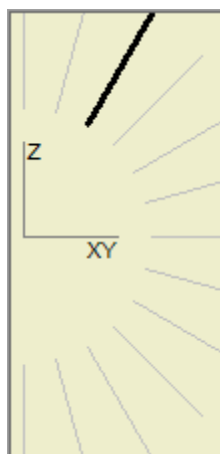
45° display mode

In *15°* or *5° mode* click in the inner area of the *Horizontal* and *Vertical* fields to choose a *45°* display mode viewpoint. Click in the outer area of the *Horizontal* and *Vertical* fields to choose another angle.

3. To set the *Vertical* view angle, do one of the following:
 - Skip this step to accept the default vertical view directions:
 - Horizontal for East, North, West and South.
 - Downwards for the other directions.

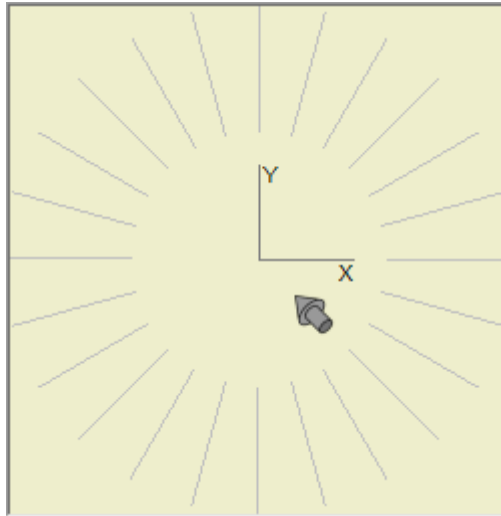
If both the down- and left-arrow show in the *Vertical* field, the default vertical view directions are active.

- Select a *45° mode* angle in the inner area, then click to confirm your choice. A single arrow indicates the selected vertical view direction.
- If in *15°* or *5° mode* in, select a view direction in the outer area, then click to confirm your choice. The selected angle displays in bold.



The view is updated as you click.

4. To set the *Horizontal* view angle, do one of the following:
 - Select a *45° mode* angle in the inner area, then click to confirm your choice.
An arrow indicates the selected horizontal view direction.
The perspective of the arrow reflects the vertical view direction.




- If in *15°* or *5° mode* in, select a view direction in the outer area, then click to confirm your choice.
The selected angle displays in bold.

The view is updated as you click.

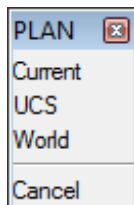
NOTE Use the *Isometric Views* toolbar for standard view rotations: top, front, back, left, right and isometric views.

Restoring Plan View

1. Do one of the following:
 - Click the *Plan View* tool button () on the *View* toolbar.
 - Choose *Plan View* in the *View* menu.
 - Type *plan* in the command bar, then press Enter.

The command bar reads: Plan view of: UCS/World/<current UCS>:

A prompt menu opens:



2. Press Enter to restore the plan view with respect to the current coordinate system.

- NOTES**
- If the WCS is the current coordinate system, the *Current* and *World* options have the same result.
 - If the *UCSFOLLOW* variable is set to *ON*, the plan view is generated whenever the UCS changes.

Layer States

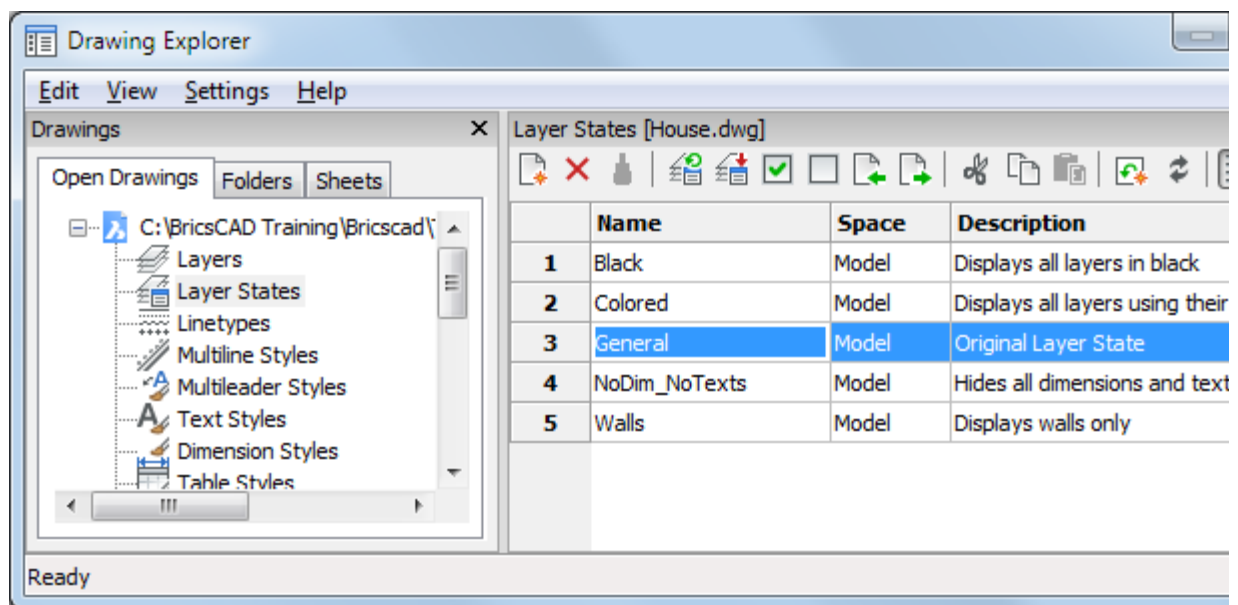
Command: LAYERSTATE



The Layer State Manager (Internet connection needed)

Layer states let you save and restore configurations of layer properties and states. You might want a layer to display in blue sometimes and green at other times, or you need some layers to be off or frozen or locked when editing a specific part of a drawing. After spending a lot of time adjusting all layer settings you can use layer states to save your settings. You can save multiple layer states in a each drawing. Layer states can be copied between drawings and you can save a layer state to an external file, which can then be imported in another drawing.

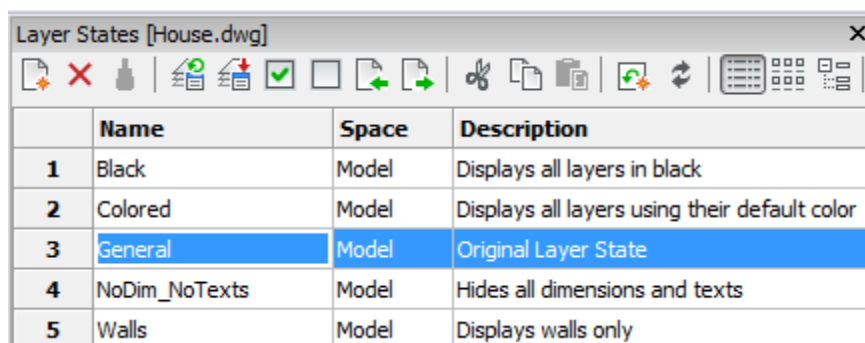
To easily create layer states, the Layer Utility commands might be useful.



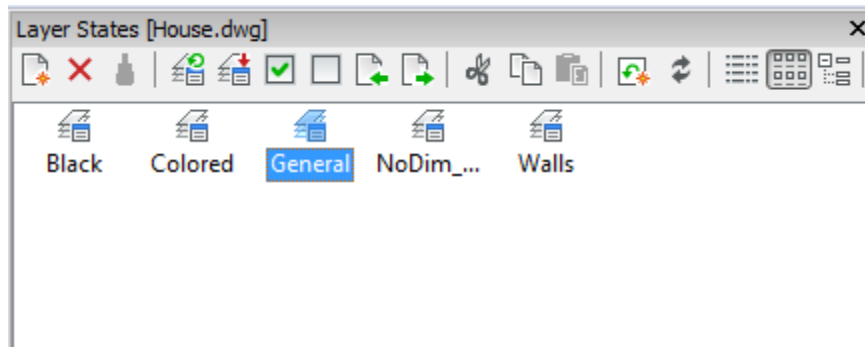
Drawing Explorer: Layer States (Detail view)


Opening the Layer States Explorer

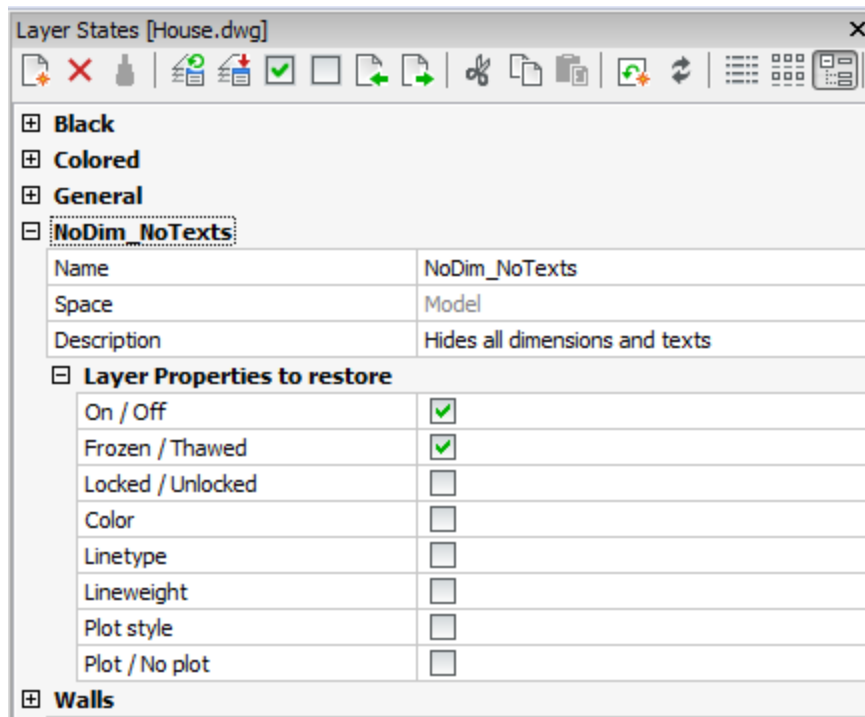
1. In the *Tools > Drawing Explorer* menu choose *Layer States...*
2. (option) Do one of the following:
 - Click the *Detail View* button (icon).





- Click the *Icon View* button ().



- Click the *Tree View* button ().





To save the current layer state

- Open the Layer States explorer.
- Click the *New* button. ().
A new layer state is created.
- Type a name for the layer state, replacing the *NewLayerState1* default name.
- (option) When in *Detail View* or *Tree View*, type a description in the *Description* field.
- (option) Click the *Tree View* button () to select the layer properties to restore.



- Expand the *Layer Properties to restore* list.

Layer Properties to restore	
On / Off	<input checked="" type="checkbox"/>
Frozen / Thawed	<input checked="" type="checkbox"/>
Locked / Unlocked	<input checked="" type="checkbox"/>
Color	<input checked="" type="checkbox"/>
Linetype	<input checked="" type="checkbox"/>
Lineweight	<input checked="" type="checkbox"/>
Plot Style	<input checked="" type="checkbox"/>
Plot / No Plot	<input checked="" type="checkbox"/>



- (option) Click the *All Properties Off* button () , then select the properties to restore.
 - (option) Click the *All Properties On* button () to select all properties.
6. Click the check box to select a layer property, click again to remove the selection.

NOTE By default in a new layer state all layer properties are selected to be restored.

To edit a layer state

1. Open the Layer States explorer.
2. (option) Click the *Overwrite* button () to overwrite the layer state with the current layer settings.
3. If necessary, click the *Tree View* button () .
4. Click the expand button of the layer state.
5. (option) Rename the layer state.
6. (option) Edit the layer state description.
7. (option) Expand the *Layer Properties to restore* list.
Click the check box to select a layer property, click again to remove the selection.



To restore a layer state

1. Open the Layer States explorer.
2. Select the layer state you want to restore.
3. Do one of the following:
 - Click the *Restore* button () .
 - Right click, then choose *Restore* in the context menu.
4. Click the *Regen* button ()


NOTE If *REGENAUTO* is on, it is not necessary to press the *Regen* button in step 4.

To copy layer states to another drawing


5. Open both, the source and the target drawing.
6. Make the source drawing the current drawing.
7. Open the Layer States explorer.

8. If the layer states display in *Tree View*, choose either *Detail View* () or *Icon View* ()
9. Do one of the following
 - Select the layer state you want to copy.
 - Press and hold the Ctrl key to select multiple layer states.
10. With the cursor on a selected layer state, press and hold the left mouse key, then drag the selection to the target drawing in the *Open Drawings* section of the Drawing Explorer dialog window.
11. When on the target drawing, release the left mouse button.
The selected layer states are copied to the target drawing.

To export a layer state

1. Open the Layer States explorer.
2. Select the layer state.
3. Click the *Export* button ()
The *Export Layer States* dialog opens.
4. In the *File Name* field, type a name for the layer state.
5. Choose a folder to save the layer state in.
6. Click the *Save* button.

To import a layer state

7. Open the Layer States explorer.
8. Click the *Import* button ()
The *Import Layer States* dialog opens.
9. Browse to the folder where you have saved the layer state file.
10. Select the layer state.
11. Click the *Open* button.
The layer state is imported.

NOTE If a layer state of the same name already exists in the drawing a warning displays. It is not possible to overwrite a layer state definition.

Named Views

Commands: -VIEW, VIEW, CAMERA

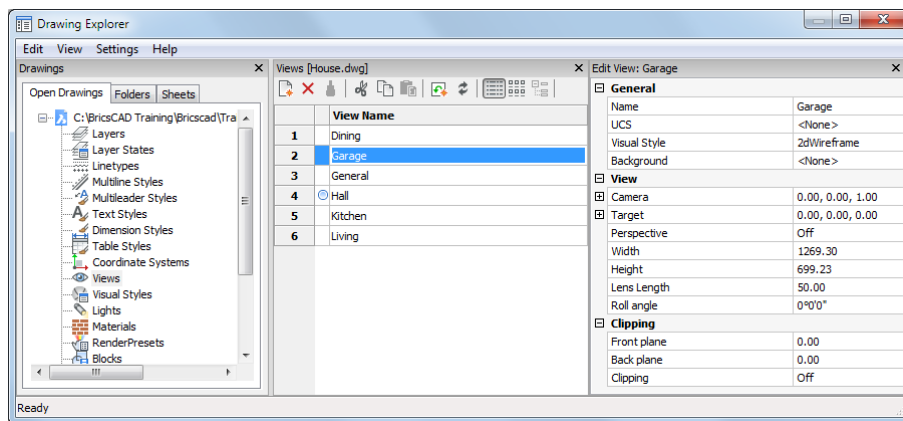
You can save the view in the current window as a named view. Saved views can be restored at any time.

A Background can be assigned to a named view, which will be used when the view is rendered.

Exploring views

- Do one of the following:
 - Choose *Drawing Explorer* > *Views...* in the *Tools* menu.
 - Type either *view* or *V* in the command bar, then press Enter.

The *Drawing Explorer - Views* window opens.
- (option) Click the blank tile in front of the *View Name* to restore a saved view. The current view is marked (●).

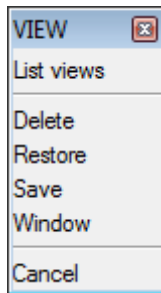


- (option) Click the *New* button (📄) in the *Drawing Explorer* toolbar to save the current view.
- (option) Click the *Delete* button (✖) in the *Drawing Explorer* toolbar to delete the selected view.
- (option) Click the *View Name*, then right click and choose *Rename* in the context menu. Type a new name for the saved view.
- (option) Check the *Perspective* property to make the view a visual perspective view. If not checked the view is a parallel perspective view. See also *Define a View*.
- (option) Click the *Background* column of a view to choose a background. The background definition of the view is applied during rendering.

Saving a view

- Do one of the following:
 - Click the *Save/Restore View* tool button (📄) on the *View* toolbar.
 - Choose *Save/Restore View* in the *View* menu.
 - Type *-view* in the command bar, then press Enter.


The command bar reads: View: ? to list saved views/Delete/Restore/Save/Window:



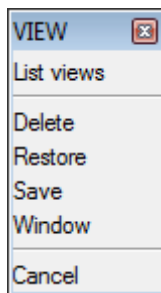
2. Do one of the following:
 - Choose *Save* in the *View* prompt menu.
 - Type *S* in the command bar, then press Enter
3. Type a name for the view in the command bar, then press Enter.
The current view is saved.

NOTE If you type a name that is already used, the current view is saved, replacing the previously saved view. There is no warning when overwriting a saved view.

Restoring a view

1. Do one of the following:
 - Click the *Save/Restore View* tool button () on the *View* toolbar.
 - Choose *Save/Restore View* in the *View* menu.
 - Type *-view* in the command bar, then press Enter.

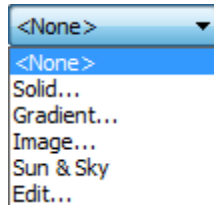
The command bar reads: View: ? to list saved views/Delete/Restore/Save/Window:



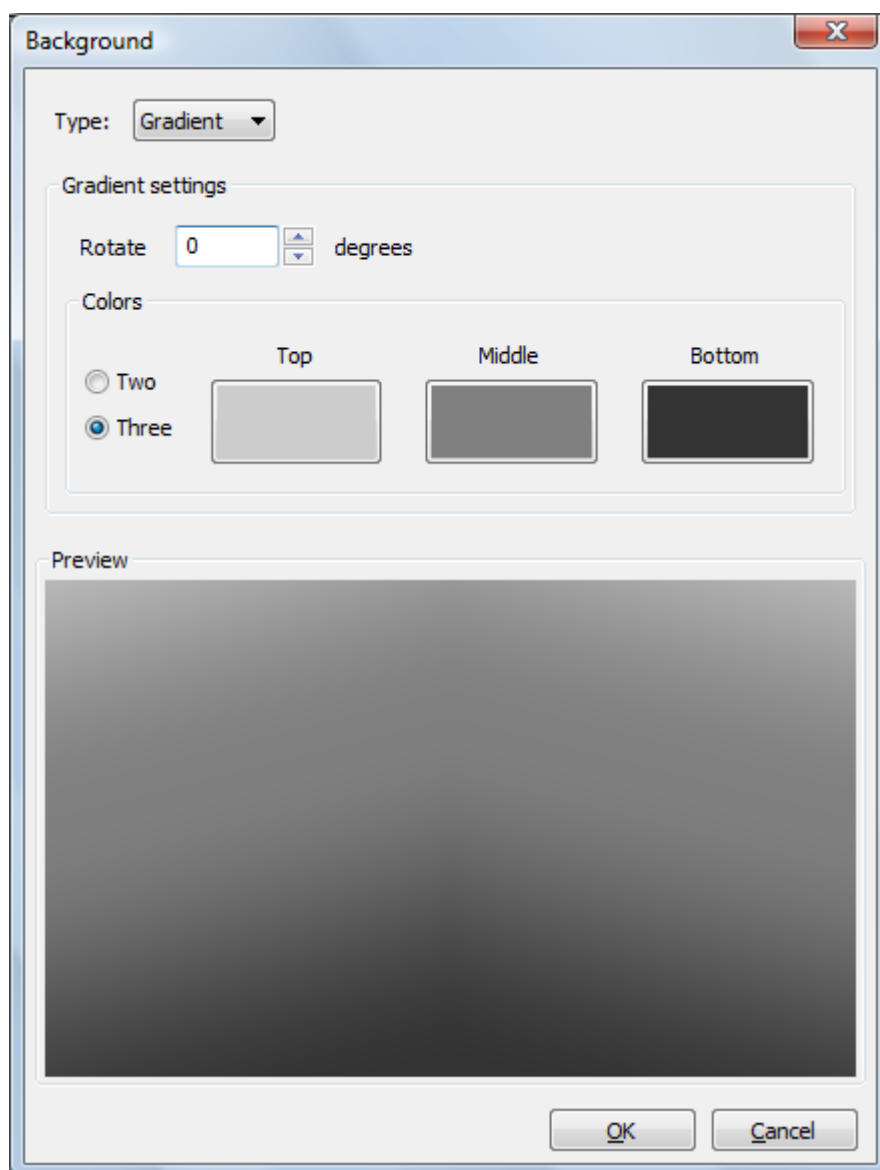
2. Do one of the following:
 - Choose *Restore* in the *View* prompt menu.
 - Type *R* in the command bar, then press Enter
3. Type the name of the saved view you want to restore in the command bar, then press Enter.
The saved view is restored.

Defining the view background

1. In the Drawing Explorer - Views dialog window, select *Background* in the *Edit* grid of the view.
2. Click the settings field to display the *Background* options.

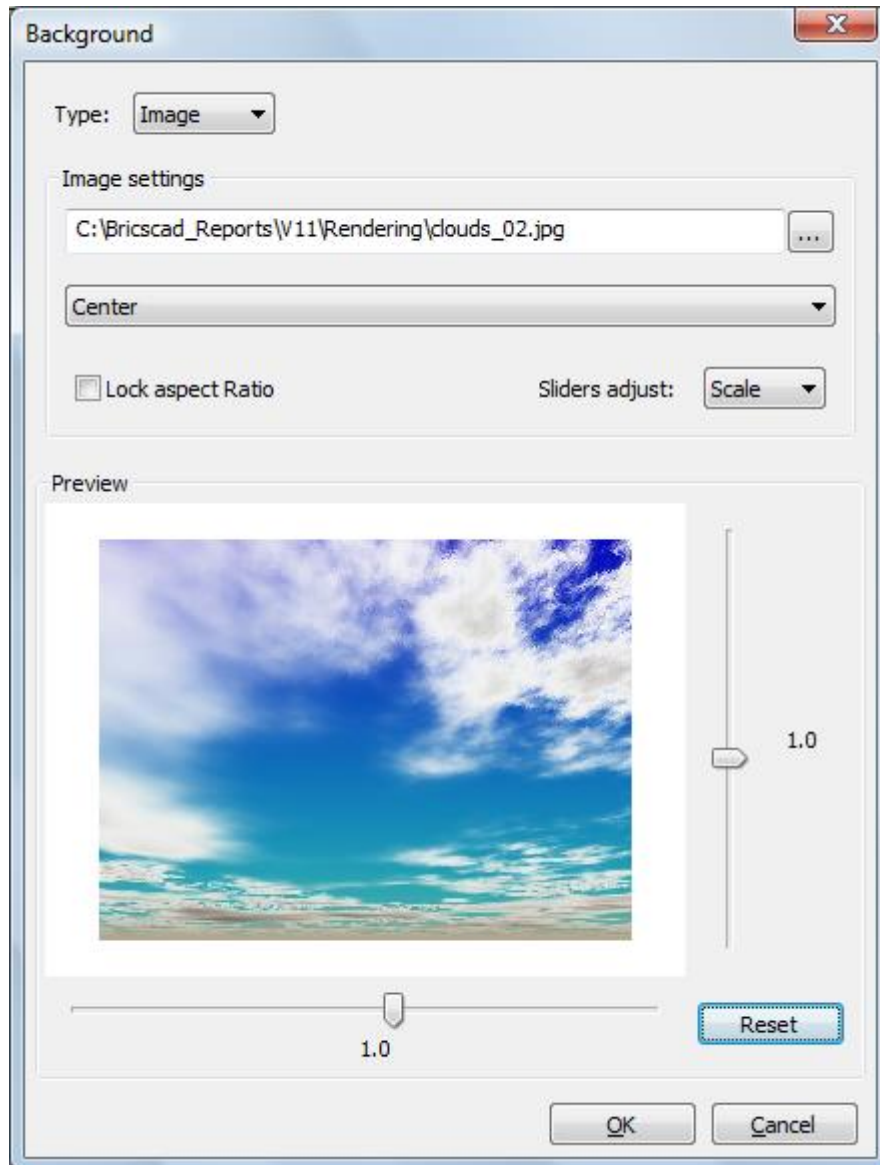


3. Choose an option in the list.
The *Background* dialog box displays:



4. (option) Select *Solid* in the *Type* list.
Click the *Color* button to define the color.
5. (option) Select *Gradient* in the *Type* list.
 - Set the rotation angle

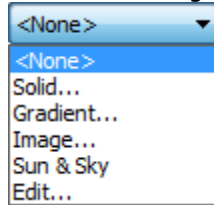
- Choose *Two* or *Three* colors.
 - Click the *Top*, *Middle* and *Bottom* color buttons to define the color.
6. (option) Select *Image* in the *Type* list.



- Click the browse button next to the Image settings field to select a raster image file.
 - Choose an alignment. The options are: *Center*, *Stretch* and *Tile*.
7. (option) Right click, then choose *Set Current* in the context menu if you want to use the newly defined background in a rendering right away.

To edit the view background

1. In the Drawing Explorer - Views dialog window, select *Background* in the *Edit* grid of the view..
2. Click the settings field to display the *Background* options.



3. Choose *Edit...* in the list to edit the current settings. The *Background* dialog box displays.
4. Edit the settings needed, then click the *OK* button.

Workspaces

Workspaces define settings and preferences regarding a specific working environment, such as 2D Drafting or 3d Modeling.

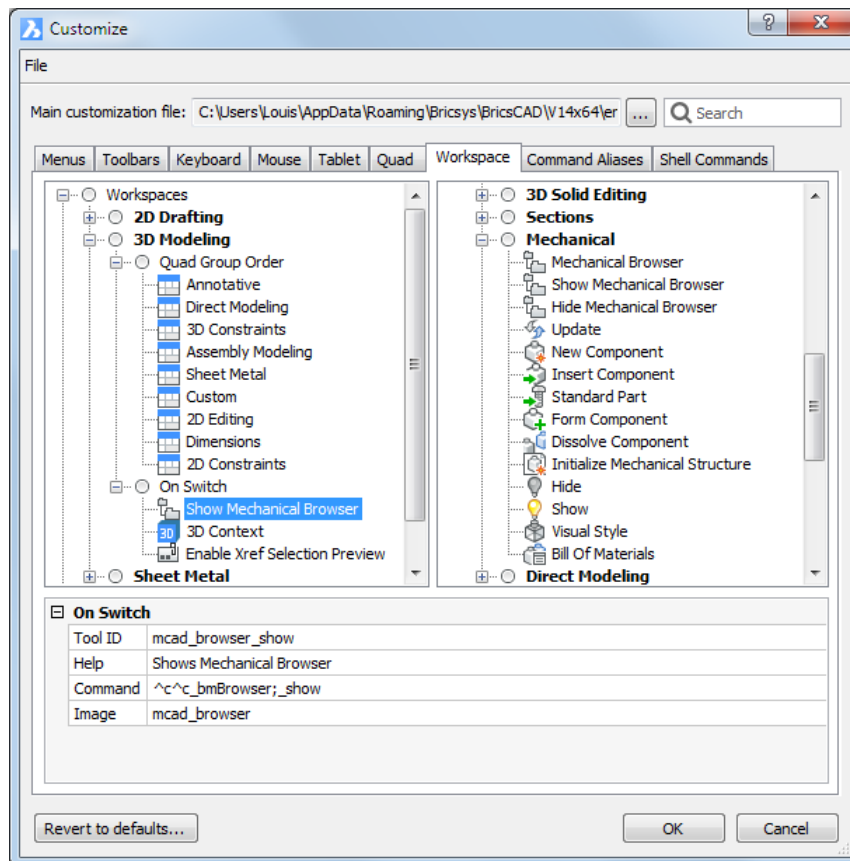
The name of the current workspace is stored in the WSCURRENT system variable.

Managing workspaces

Workspaces are defined in the *Workspace* tab page on the *Customize* dialog.

Do one of the following:

- Open the *Customize* dialog, then select the *Workspace* tab.
- Right click the *Current Workspace* field in the *Status Bar*, then choose *Customize Workspaces...* in the context menu.



3 workspaces are available in the *Default.cui*:

- 2D Drafting
- 3D Modeling
- Sheet Metal

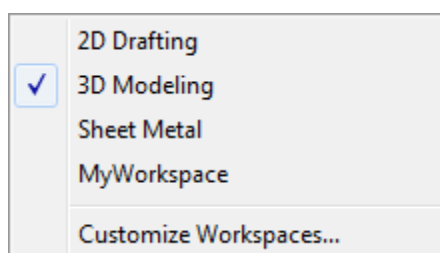
A workspace definition is composed of:

- The *Quad Group Order*, which defines the order and availability of the Quad command groups.
- The *On Switch* sequence, which defines the system variables and user preferences to be set and the commands to be executed when switching to the workspace.

Setting the current workspace

Do one of the following:

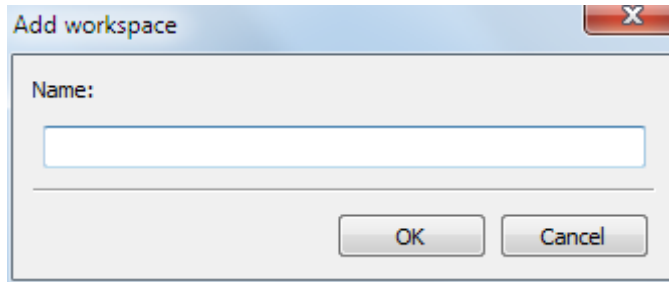
- Right click the *Current Workspace* field in the Status Bar, then select a workspace in the context menu.



- Type *wscurrent* at the command prompt.
The command bar reads: New value for WSCURRENT, or . for none/<"2d DRAFTING">:
Type the name of the new current workspace, then press enter.

Creating a workspace

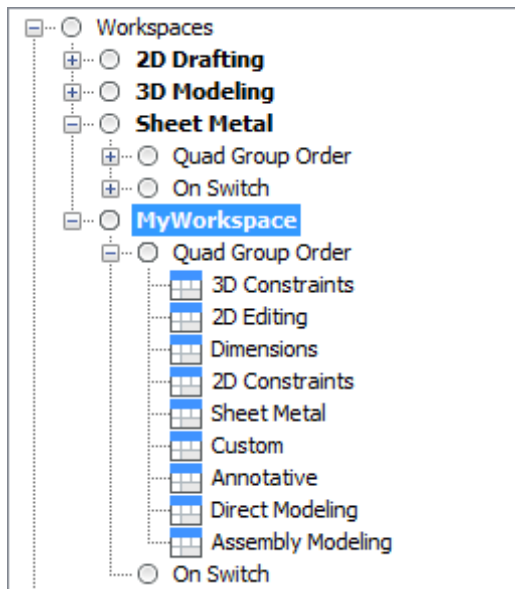
1. Right click the *Current Workspace* field in the Status Bar, then choose *Customize Workspaces...* in the context menu.
2. Do one of the following:
 - Select *Workspaces*, then right click and choose *Append Workspace* in the context menu.
 - Select an existing workspace, then right click and choose *Insert Workspace* in the context menu.
3. Type a name for the new workspace, then click the OK button.



When appending a workspace, the new workspace is created below the last existing workspace.

When inserting a workspace, the new workspace is created above the selected workspace.

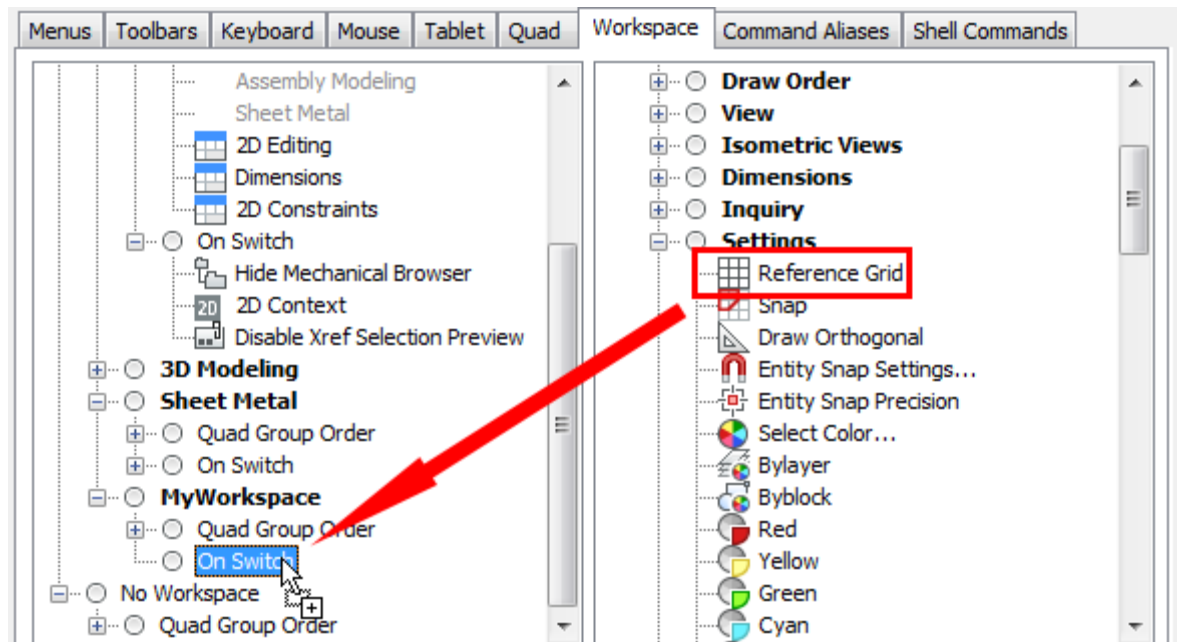
A new workspace contains a *Quad Group Order* list and an empty *On Switch* sequence.



Editing a workspace

1. Right click the *Current Workspace* field in the Status Bar, then choose *Customize Workspaces...* in the context menu.
2. If necessary, click the *Expand* button (+) of the workspace.
3. (option) To change the order of the Quad command groups: press and hold the left mouse button, then drag a command group to its new position.

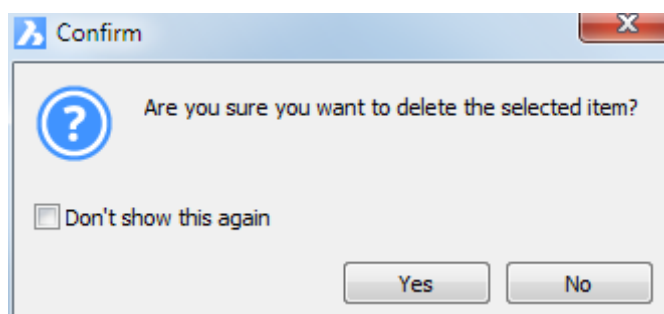
4. (option) Right click a Quad command group, then choose *Switch off* in the context menu to remove the command group from the Quad cursor menu.
5. (option) Right click a currently switched off Quad command group then choose *Switch on* in the context menu to add the command group to the Quad cursor menu.
6. (option) Add a command to the *On Switch* sequence: select a command in the available commands pane and drag it to the On Switch sequence.



NOTE See To create a new tool if you want to add a custom tool to the *On Switch* sequence.

Deleting a workspace

1. Right click the *Current Workspace* field in the Status Bar, then choose *Customize Workspaces...* in the context menu.
2. Right click the workspace, then choose *Delete* in the context menu. A Confirm dialog box displays:



3. Press the Yes button to delete the selected workspace.

Define a View

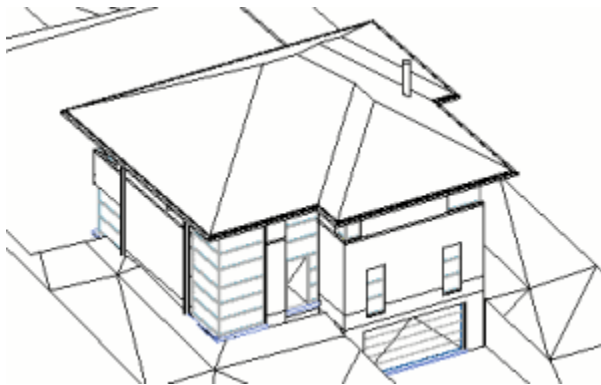
Commands: DVIEW and CAMERA

The *Define View* command dynamically defines parallel perspective or visual perspective views of your 3D models.

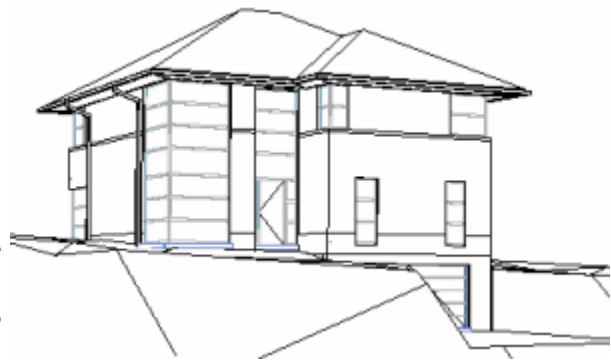
The *Camera* command creates perspective views.

The projection method of a view is controlled by the *Perspective* property of the view:

- If ON, the view is a visual perspective view.
- If OFF, the view is a parallel perspective view.



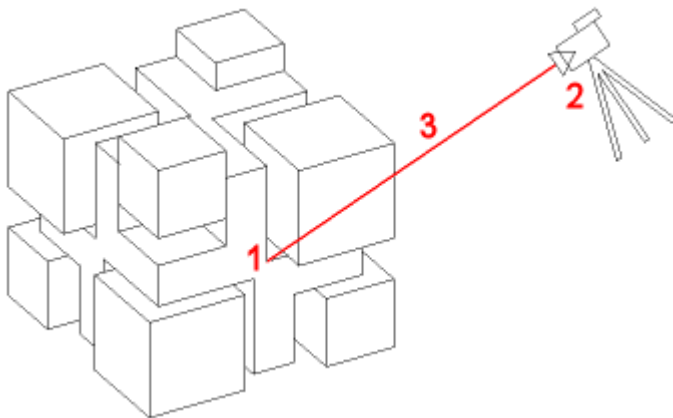
Parallel Perspective View



Visual Perspective View

The following parameters are used in the *Define View* procedure:


- *Target Point*: Defines the center of the view (1)
- *Camera Point*: Defines the view point and view direction (2).
- *Distance to Target*: Distance between the camera position and the target point (3).



NOTE View manipulation tools, such as zoom, pan, real-time motion, are not available in visual perspective views. However, you can use the mouse to manipulate the view.

Defining a camera view

The *Camera* command creates a named view.

1. Do one of the following:
 - Click the *Camera* tool button () on the *View* toolbar.
 - Choose *Camera* in the *View* menu.
 - Type *Camera* in the command bar.

The command bar reads: Specify camera location:
2. Do one of the following:
 - Click a point in the drawing.
 - Type the coordinates of the camera location in the command bar and press Enter.


The command bar reads: Specify target location:
3. Do one of the following:
 - Click a point in the drawing.
 - Type the coordinates of the target location in the command bar and press Enter.

The command bar reads: Enter an option:
 ?/Name/Location/Height/Target/Lens/Clipping/View/<eXit>:
4. Do one of the following:
 - Press Enter to create the camera view using a default name (e.g. Camera1).
 - Enter an option:
 - **Name:**
 Prompts you: Enter name for new camera <Camera1>:
 Type a name at the command prompt, then press Enter.
 - **Location:**
 Prompts you: Specify camera location <current location>:
 Click a point or enter new coordinates of the camera location, then press Enter.
 - **Height:**
 Prompts you: Specify camera height <current height>:
 Enter a new height (Z-coordinate) for the camera location.
 - **Target:**
 Prompts you: Specify target location <current location>:
 Click a point or enter new coordinates of the target location, then press Enter.
 - **Lens:**
 Prompts you: Specify lens length in mm <50.00>:
 Enter a new lens length, then press Enter.
 - **Clipping:**
 Prompts you: Enable front clipping plane? <Yes>/No:
 Choose Yes to define the front clipping plane.
 Prompts you: Specify front clipping plane offset from target plane <0.00>:
 Type value in the command bar or specify the distance by clicking two points.

 Prompts you: Enable back clipping plane? Yes/<No>:
 Choose Yes to define the front clipping plane.
 Prompts you: Specify front clipping plane offset from target plane <0.00>:
 Type value in the command bar or specify the distance by clicking two points.
 - **View:**
 Prompts you: Switch to camera view? Yes/<No>:
 Choose Yes to display the camera view.
 This option concludes the *Camera* command.

NOTE Choose *Drawing Explorer > Views...* in the *Tools* menu to edit a camera view.

To toggle the perspective property of view

1. Do one of the following
 - Click the *Perspective* tool button () on the *View* toolbar.
 - Choose *Perspective* in the *View* menu.
 - Type *perspective* in the command bar.

The command bar reads: New current value for PERSPECTIVE (Off or On) <On>:
A prompt menu displays.

2. Do one of the following:
 - Type *off* or *on* in the command bar, then press Enter.
 - Choose *Off* or *On* in the prompt menu.

NOTE In named views you can set the *Perspective* property in the Drawing Explorer - Views dialog.

Model space and paper space

Understanding paper space and model space

When you start a drawing session, your initial working area is called *Model Space*. Model Space is an area in which you create two-dimensional and three-dimensional entities based on either the World Coordinate System (WCS) or a user coordinate system (UCS). You view and work in model space while using the *Model* tab.

In general model space consists of a single view that fills the screen. If needed, you can create additional views, called viewports, which can show different views of your drawing or 3D model. All viewports are displayed in a tiled manner. You can work in only one of these viewports at a time but all viewports are updated simultaneously. Click in a viewport to make it the current viewport. You can print the current viewport only. The Viewports command lets you manage your viewports in model space.

BricsCAD provides an additional work area, called *Paper Space*. Paper space represents a paper layout of your drawing. In this work area, you can create and arrange different views of your drawing similar to the way you arrange drawings on a sheet of paper. In paper space you can also add keynotes, annotations, borders, title blocks, and other print-related entities, which you don't want to see in model space.

Each drawing has at least one *Layout* in which you can have one or more *Layout Viewports*. Such layout viewports are to be considered as a view window on your drawing in model space. Layout viewports can be placed anywhere in a paperspace layout. Each viewport has its own scale and layer visibility. All viewports in the same layout can be printed simultaneously.

The Mview command lets you manage your viewports in paper space.

Although not necessary to print your drawing, paper space offers a lot of advantages:

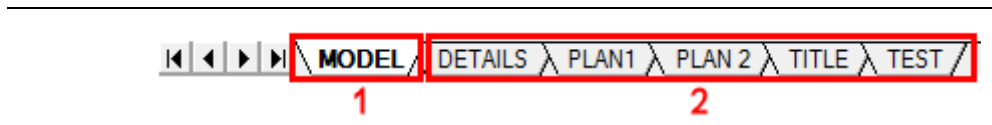
- Create multiple layouts to print the same drawing with different print settings, such as pen widths, printer configuration files, lineweight settings, drawing scale, and more.
- Add print-related entities that are not essential to the model itself, such as keynotes, annotations, title blocks, etc.
- For a single layout, create multiple layout viewports to print multiple views of your drawing at different scales.

Switching between model space and paper space

- To switch between Model Space and Paper Space Layouts use the Model and Layout tabs at the bottom of the drawing window.
- When working in a layout, use the MSPACE and PSPACE commands to toggle between paper space and model space.

Using the model and layout tabs

1. (option) To open model space, click the *Model* tab (1) at the bottom of the drawing window.
2. (option) To open a paper space layout, click the corresponding *Layout* tab (2) at the bottom of the drawing window.



NOTE A drawing contains at least one layout, which is named *Layout1* by default.

Toggle between model space and paper space in a layout

Commands: MSPACE and PSPACE

1. (option) When in *model space* (Model Space, with floating viewports) do one of the following to switch to *paper space*:
 - Type either *pspace* or *ps*, then press Enter
 - Double click outside a viewport.
2. (option) When in *paper space* do one of the following to switch to *model space* (Model Space, with floating viewports):
 - Type either *mpace* or *ms*, then press Enter.
 - Double click inside a viewport.

NOTE *MSPACE* and *PSPACE* commands are available when working in a layout only.

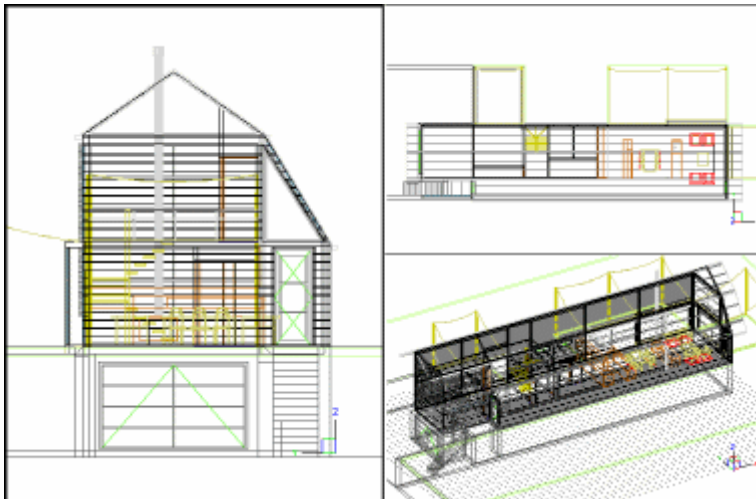
Model Space Viewports

Command: VPORTS

The display in model space (Model Space, with tiled viewports) can be divided into multiple viewports, each of which can contain a different view of the current drawing. All viewports are displayed in a tiled manner. You can work in only one of these viewports at a time but all viewports are updated simultaneously. When no command is active click in a viewport to make it the current viewport. You can print the current viewport only.

The following settings can be defined differently for each viewport:


- grid display and snap
- coordinate system: WCS or UCS



Model Space divided in 3 viewports

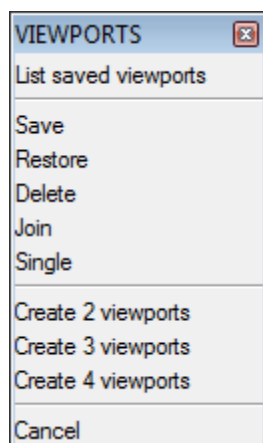
Creating viewports in model space

1. Do one of the following:

- Click the *Viewports* tool button () on the *Views* toolbar.
- Choose *Viewports* in the *Views* menu.
- Type *vports* in the command bar.

The command bar reads: Viewports: ? to list/Save/Restore/Delete/Single/Join/2/3/4/<3>:



A prompt menu opens.



2. (option) To create 2 viewports, do one of the following:

- Choose *Create 2 viewports* in the prompt menu.
- Type 2 in the command bar, then press Enter.

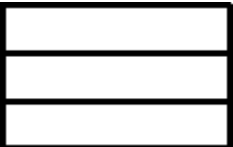

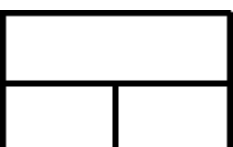
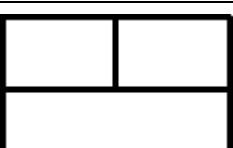
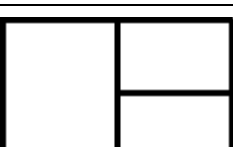
The command bar reads. Two viewports: Horizontal/<Vertical>:

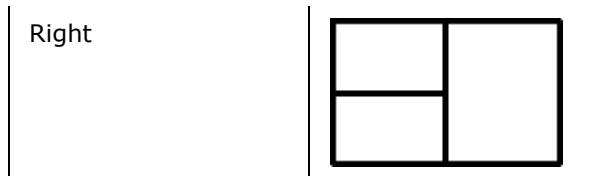
Option	Result
Horizontal	
Vertical	

3. (option) To create 3 viewports, do one of the following:

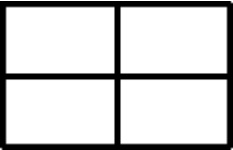
- Choose *Create 3 viewports* in the prompt menu.
- Type 3 in the command bar, then press Enter.

The command bar reads. Three viewports:
Horizontal/Vertical/Above/Below/Left/<Right>:

Option	Result
Horizontal	
Vertical	
Above	
Below	
Left	



4. (option) To create 4 viewports, do one of the following:
 - Choose *Create 4 viewports* in the prompt menu.
 - Type *4* in the command bar, then press Enter.

Option	Result
4	

Drawing in multiple viewports

1. Click in a viewport to make it current.
The border of the current viewport highlights.
2. Draw the entities in the current viewport.
3. (option) Repeat step 1 to make a different viewport current.

- NOTES**
- The crosshairs show in the current viewport only.
 - You can start a command in the current viewport, and finish the operation in a different viewport.

Joining adjacent viewports

1. Launch the *Viewports* command.
2. Select the *Join* command option.
The command bar reads: Select inside dominant viewport <Current>:
3. Click in the dominant viewport or press Enter to choose the current viewport.
The border of the selected viewport highlights.
The command bar reads. Select inside viewport to join:
4. Click the viewport to join again to confirm.
The two selected viewports are joined.

NOTE You can join viewports which share an edge of equal length only.

Paper space viewports

Commands: MVIEW, MVSETUP, ALIGNSPACE and VPCLIP

The *Mview* command creates one or more viewports in layout tabs (short for "make viewports").

The *MvSetup* command creates multiple viewports; in model space the command creates a rectangle, representing a paper sheet with respect to a specified scale.

The *AlignSpace* command adjusts viewport angle, zoom factor and pan position based on alignment points specified in model space and paper space.

The *Vpclip* command clips viewports in layouts (short for "view port clipping").

In a layout you can create multiple viewports each of which displays a unique view of the entities created in model space. Each layout viewport functions as a window into your model space drawing. You can control the view, scale, and content of each layout viewport separately.

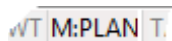
A layout viewport is created as a separate entity that you can copy, delete, move, scale, and stretch as you would any other drawing entity. You can snap to the viewport borders using entity snap. When you are working in model space with floating viewports (see Toggle between model space and paper space), click any layout viewport to make it the current viewport, and then add or modify model space entities in that viewport. Any changes you make in one layout viewport are immediately visible in the other viewports (if the other layout viewports are displaying that portion of the drawing). Zooming or panning in the current viewport affects only that viewport.

Each viewport has its own layer visibility settings. You can also turn off the display of the content of a viewport.

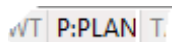
To preserve the scaling of a viewport you can lock the display. It is no longer possible to zoom or pan in a locked viewport.

Apart from viewports, you can add print-related entities in a paper space layout that are not essential to the model itself, such as keynotes, annotations, title blocks, etc. Such entities are part of a specific paper space layout and do not appear in other layouts or in model space.

When you are working in a layout, either *Model Space* or *Paper Space* is your current workspace. The *Workspace* field in the Status Bar indicates which workspace is current: *M: <Layout Name>* indicates you are working in *Model Space*, while *P: <Layout Name>* means *Paper Space* is the current workspace.

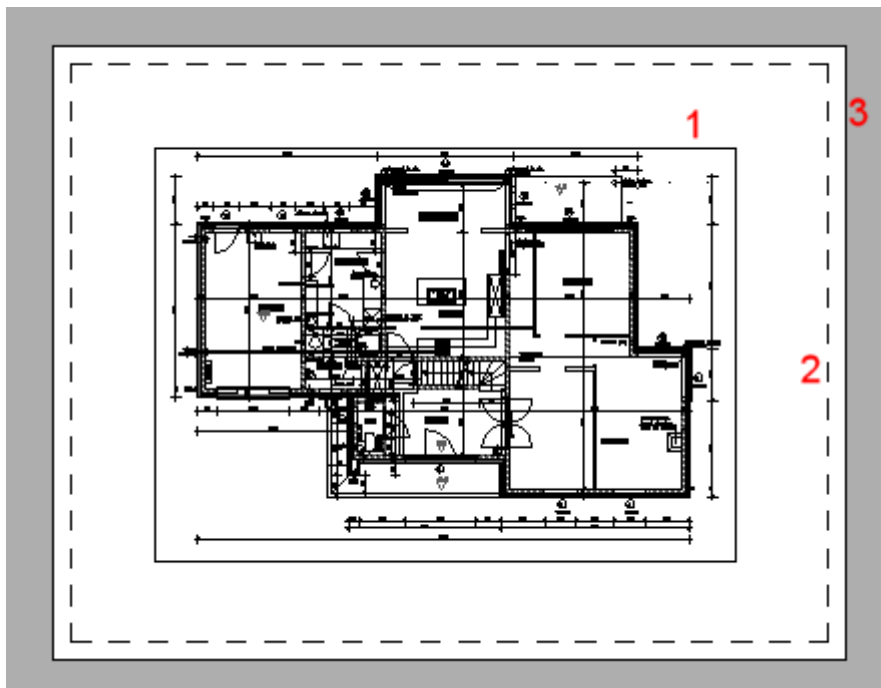


Model Space, with floating viewports of layout PLAN



Paper Space of layout PLAN

Displaying the paper sheet and the printable area



1. Viewport
2. Printable area.
The display of the printable area is controlled through the DISPPAPERMARGINS system variable.
3. Paper sheet.
The display of the paper sheet is controlled through the DISPPAPERBKG system variable.

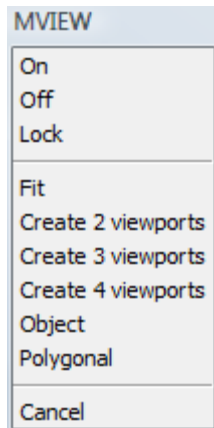
Creating viewports in a layout

Command: MVIEW

1. Click the appropriate layout tab at the bottom of the drawing window.
2. Do one of the following:
 - Click the *Paper Space Views* tool button (📄) on the *Views* toolbar.
 - Choose *Paper Space Views* in the *Views* menu.
 - Type *mview* at the command prompt, then press Enter.

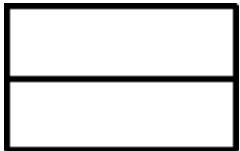

The command bar reads: Viewports. ON/OFF/Fit/2/3/4/<First corner>:

A prompt menu opens:



3. (option) To add 1 viewport. do one of the following:
 - Click to specify the first corner of the viewport, then specify the opposite corner.
A single viewport which fits in the specified rectangle is created.
 - Choose *Fit to view* in the prompt menu or type *F*, then press Enter.
A single viewport which fits in the current drawing display window is created.
4. (option) To create 2 viewports, do one of the following:
 - Choose *Create 2 viewports* in the prompt menu.
 - Type 2 in the command bar, then press Enter.

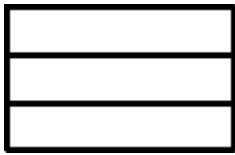

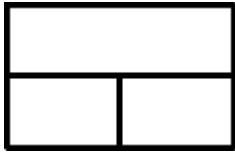
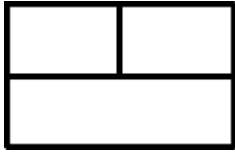
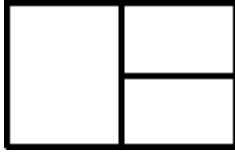

The command bar reads. Two viewports: Horizontal/<Vertical>:

Option	Result
Horizontal	
Vertical	

The command bar reads: Fit to screen/<First corner of bounding rectangle>:

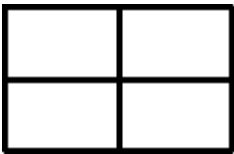
- Click to specify the first corner of the bounding rectangle, then specify the opposite corner.
Two viewports which fit in the specified bounding rectangle are created.
 - Choose *Fit to view* in the prompt menu or type *F*, then press Enter.
Two viewports which fit in the current drawing display window are created.
5. (option) To create 3 viewports. do one of the following:
 - Choose *Create 3 viewports* in the prompt menu.
 - Type 3 in the command bar, then press Enter.

The command bar reads. Three viewports:
Horizontal/Vertical/Above/Below/Left/<Right>:

Option	Result
Horizontal	
Vertical	
Above	
Below	
Left	
Right	

The command bar reads: Fit to screen/<First corner of bounding rectangle>:

- Click to specify the first corner of the bounding rectangle, then specify the opposite corner.
Three viewports which fit in the specified bounding rectangle are created.
 - Choose *Fit to view* in the prompt menu or type *F*, then press Enter.
Three viewports which fit in the current drawing display window are created.
6. (option) To create 4 viewports. do one of the following:
- Choose *Create 4 viewports* in the prompt menu.
 - Type *4* in the command bar, then press Enter.

Option	Result
Create 4 viewports	

The command bar reads: Fit to screen/<First corner of bounding rectangle>:

- Click to specify the first corner of the bounding rectangle, then specify the opposite corner.
Four viewports which fit in the specified bounding rectangle are created.
 - Choose *Fit to view* in the prompt menu or type *F*, then press Enter.
Four viewports which fit in the current drawing display window are created.
7. (option) To create a non-rectangular (clipped) viewport using an existing polyline or circle, do one of the following:
- Choose *Object* in the prompt menu.
 - Type *O* in the command bar, then press Enter.
- The command bar reads: Select Object to clip viewport.
- Select a closed polyline or a circle in the layout.
8. (option) To create a non-rectangular (clipped) viewport, do one of the following:
- Choose *Polygonal* in the prompt menu.
 - Type *P* in the command bar, then press Enter.
- The command bar reads: Specify start point:
- Specify the vertices of the non-rectangular viewport. Right click or press Enter to create the viewport.

NOTES

- Make sure no other viewports display when choosing the *Fit to view* option because they will be hidden behind the newly created viewport.
- Each newly created viewport shows all entities on the layers which are currently visible in model space.
- Since a new viewport is created on the current layer it is recommended to use a dedicated layer for viewport entities. Make this layer current when you want to add a new viewport. If you don't want the viewport outlines to be printed, set the *Do Not Plot* property of the viewport layer.
- Only circles and closed polylines can be used to create a clipped viewport. If you want to create an elliptical viewport, use the Polyline Ellipse variable to create a polyline representation of an ellipse.
- To select a clipped viewport do the following: Click the viewport border. Both the viewport and the clipping polygon are selected then, as indicated in the Properties Bar: the list button caption reads: *All (2)*. Press the list button and select *Viewport* to see the properties of the viewport.

Creating an array of layout viewports

Command: MVSETUP

1. Type *mvsetup* in the command bar.
The command bar reads: Enter an option [Align/Create/Scale viewports/Undo]:
2. Do one of the following:
 - Type *C* in the command bar, then press Enter.
 - Choose *Create* in the context menu.

The command bar reads: Enter an option [Delete objects/Create viewports/Undo] <Create>:
3. Press Enter to accept the default option.
The command bar reads: Enter choice for layout options <0-3> [none(0)/single(1)/std. engineering(2)/array of viewports(3)] <0>:
4. Do one of the following:
 - Type *3* in the command bar, then press Enter.
 - Choose *Array of Viewports* in the prompt menu.

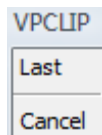
The command bar reads: Specify first corner of bounding area for viewport(s):

5. Pick a point in the layout.
The command bar reads: Other corner of rectangle:
6. Pick a second point in the layout.
The command bar reads: Enter number of viewports in X direction <1>:
7. Specify the number columns in the array.
The command bar reads: Enter number of viewports in Y direction <1>:
8. Specify the number of rows in the array.
The command bar reads: Specify distance between viewports in X direction <0.000>:
9. Do one of the following:
 - Press Enter, to create contiguous viewport columns.
 - Specify the horizontal distance between the viewports.
 The command bar reads: Specify distance between viewports in Y direction <horizontal distance>:
10. Do one of the following:
 - Press Enter to set the vertical spacing equal to the horizontal spacing.
 - Type 0 (zero) to create contiguous viewport rows.
 - Specify the vertical spacing between the viewports.
 The viewports are created.
11. Press Enter to conclude the command.

To clip a viewport

Command: VPCLIP

1. Type *vpclip* in the command bar, then press Enter.
The command bar reads: Select viewport to clip:
2. Select the viewport.
The command bar reads: Polygonal <Select clipping object>:
A prompt menu displays.



3. Do one of the following:
 - Select the clipping object.
 - Choose *Polygonal* in the prompt menu or type *P*, then press Enter.
You are prompted to draw a polyline.

The viewport is clipped.

NOTE

- Only closed polylines and circles are accepted as clipping boundaries.
- If you want elliptical viewports, use the polyline representation to create the ellipses.

To remove the clipping boundary of a viewport

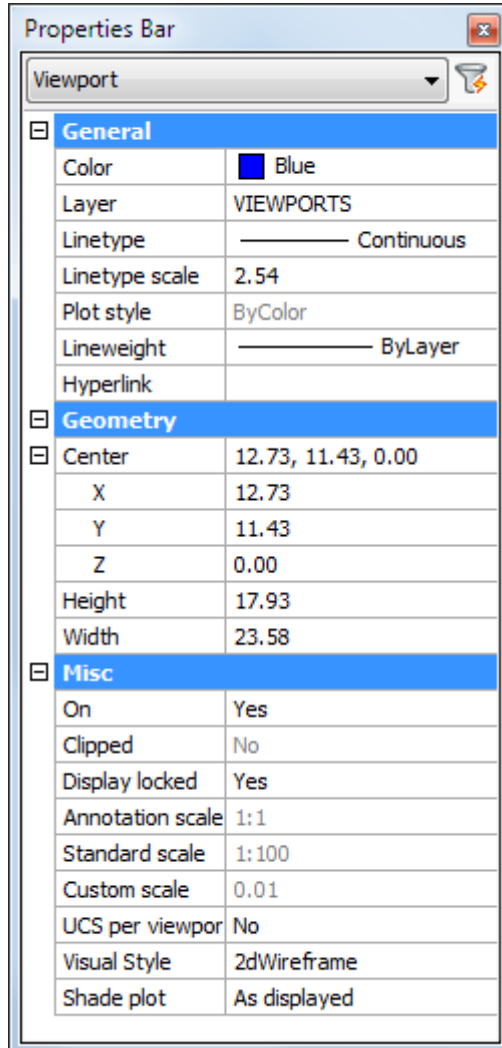
Command: VPCLIP

1. Type *vpclip* in the command bar, then press Enter.
The command bar reads: Select viewport to clip:

2. Select a clipped viewport.
The command bar reads: Polygonal/Delete<Select clipping entity>:
3. Do one of the following:
 - Type *D*, then press Enter.
 - Choose *Delete* in the prompt menu.

Defining viewport properties

1. Select the viewport.
The viewport properties display in the Properties Bar.



Properties Bar	
Viewport	
General	
Color	Blue
Layer	VIEWPORTS
Linetype	Continuous
Linetype scale	2.54
Plot style	ByColor
Lineweight	ByLayer
Hyperlink	
Geometry	
Center	12.73, 11.43, 0.00
X	12.73
Y	11.43
Z	0.00
Height	17.93
Width	23.58
Misc	
On	Yes
Clipped	No
Display locked	Yes
Annotation scale	1:1
Standard scale	1:100
Custom scale	0.01
UCS per viewport	No
Visual Style	2dWireframe
Shade plot	As displayed

2. Click the settings field of a property to modify.

Properties

Name	Description
Center	XYZ coordinates of the center point of the viewport. To specify the center of the viewport graphically, drag the center handle of the viewport.
Height (*)	Height of the viewport in <i>drawing units</i> . To specify the height of the viewport graphically, drag the top or bottom handle of the viewport.
Width (*)	Width of the viewport in <i>drawing units</i> . To specify the width of the viewport graphically, drag the left or right handle of the viewport.
On	Controls the display of the content of the viewport.
Clipped	Allows to create non-rectangular viewports.
Display locked	Locks the scaling of the viewport content to preserve the scaling factor.
Standard scale (*)	Lets you choose a standard scaling factor. The SCALELISTEDIT command allows to edit the available scales in the current drawing.
Custom scale (*)	Lets you define the scaling factor in a decimal format. Displays the current scaling factor.
UCS per viewport	If Yes, lets you define a UCS for this viewport.
Shade Plot	Defines how the viewport will be plotted. The options are: <div> As displayed Legacy wireframe Legacy hidden Rendered 2dWireframe 3dWireframe 3D Hidden Realistic Conceptual High Quality Modeling </div>

(*) You must choose the width and height of a paperspace viewport with respect to the viewport scale.

Navigating in a paper space layout

View manipulation commands, such as zoom, pan and view manipulation using the mouse, act slightly different in a paper space layout compared to model space (with tiled viewports).

If you are working in paper space (no viewport selected), view manipulation commands act on the complete paper space layout.

If you are working in a viewport (model space with floating viewports), view manipulation commands act on the active viewport only. Except if the display of the current viewport is locked, then the view manipulation commands act on the complete paper space layout.

NOTE It is not possible to rotate the display of a paper space layout.
View Rotate commands are available in viewports of which the display is not locked only.

Setting the layer visibility in a viewport

1. Do one of the following:
 - If *Paper Space* is the current workspace: double click inside the viewport to make it the current viewport.
 - If *Model Space with floating viewports* is the current workspace: click inside the viewport to make it the current viewport.
2. Do one of the following.
 - Click the *Layers...* tool button on the *Settings* toolbar.
 - Choose *Layers...* in the *Settings* menu.
 - Type *layer* in the command bar, then press Enter.

The *Drawing Explorer - Layers* window opens.
3. In the *Curr. VP* (Current Viewport) column:
 - click the frozen icon (❄) of the layer(s) you want to thaw.
 - click the thawed icon (☀) of the layer(s) you want to freeze.
4. Close the *Drawing Explorer - Layers* window.
5. (option) Repeat steps 1 through 4 to set the layer visibility in another viewport.

NOTE Use the *Viewports / Vpfreeze* option of the LAYOFF and LAYFRZ commands to freeze layers in a viewport by clicking entities.

Setting the scale of a viewport

1. Make sure the Plot Scale property of the layout is set correctly.
2. Switch to *paperspace*.
(See Toggle between model space and paper space)
3. Click the viewport border.
The viewport properties display in the BricsCAD Properties Bar.
4. Do one of the following:
 - Choose a scale in the *Standard scale* list.
 - Type a scaling factor in the *Custom scale* field.
5. (option) If necessary, adjust the size of the viewport to the new scale.
6. (option) Set the *Display Locked* property to *Yes*.

NOTE In a *locked viewport* it is impossible to:

- zoom, pan or rotate the view
- modify the scale

Setting the scale of a selection of viewports

Command: MVSETUP

1. Make sure the Plot Scale property of the layout is set correctly.
2. Type *mvsetup* in the command bar.
The command bar reads: Enter an option [Align/Create/Scale viewports/Undo]:
3. Do one of the following:
 - Type *S* in the command bar, then press Enter.
 - Choose *Scale viewports* in the prompt menu.
 The command bar reads: Select entities
4. Select the viewports.
5. Press Enter or right click to stop selecting viewports.
The command bar reads: Set the scaling mode of viewports:
Interactively/<Uniform>:
6. Do one of the following:
 - Press Enter to set the scale of the selected viewports uniformly.
The command bar reads: Enter the number of paper space units <1.0>:
 - Press Enter to accept the number of paper space units.
The command bar reads: Enter the number of model space units <1.0>:
 - Type the desired scaling factor: e.g. 50 to apply a 1/50 scale.
 - Press Enter to apply the scale to all selected viewports.
 - Press Enter to conclude the command.
 - Choose *Interactively* in the prompt menu or type *I*, then press Enter.
The first viewport in the selection set highlights.
The command bar reads: Enter the number of paper space units <1.0>:
 - Press Enter to accept the number of paper space units.
The command bar reads: Enter the number of model space units <1.0>:
 - Type the desired scaling factor: e.g. 50 to apply a 1/50 scale.
 - Press Enter to apply the scale to the viewport.
The next viewport highlights.
The command bar reads: Enter the number of paper space units <1.0>:
 - Repeat the previous steps for each of the selected viewports.
7. Press Enter to conclude the command.

To rotate the display in a viewport


The display of a layout viewport can be rotated only if the display of the viewport is not locked.

If the VPROTATEASSOC system variable is ON, the ROTATE command rotates the content of a viewport over a specified angle.

The *Align > Rotate* option of the MVSETUP command rotates the content of a viewport relative to the X-axis of the world coordinate system (WCS).

Rotating the viewport display

1. Select the viewport, then right click and choose *2D Rotate* in the context menu.
The command bar reads: Rotation point.

2. Use the *Center* () entity snap to select the center point of the viewport as the rotation point.
The command bar reads: Copy/Base angle/<Rotation angle> <0>:
3. Type the rotation angle, then press Enter.
4. The content of the viewport is rotated over the specified angle.

Defining the rotation angle of the viewport display

1. Type *mvsetup* in the command bar.
The command bar reads: Enter an option [Align/Create/Scale viewports/Undo]:
2. Do one of the following:
 - Type *A* in the command bar, then press Enter.
 - Choose *Align* in the prompt menu.

The command bar reads: Enter an option [Angled/Horizontal alignment/Vertical alignment/Rotate view/Undo]:
3. Do one of the following:
 - Type *R* in the command bar, then press Enter.
 - Choose *Rotate view* in the prompt menu.

The command bar reads: Specify basepoint in the viewport to be rotate:
4. Specify the rotation point in the viewport.
The command bar reads: Specify rotation angle:
5. Type the rotation angle in the command bar, then press Enter.
The viewport display is rotated over the specified angle with respect to the X-axis of the WCS.

NOTE Type 0 (zero) in the final step of the above procedure to undo the rotation of the viewport display.

To align viewports

Command: MVSETUP

1. Type *mvsetup* in the command bar.
The command bar reads: Enter an option [Align/Create/Scale viewports/Undo]:
2. Do one of the following:
 - Type *A* in the command bar, then press Enter.
 - Choose *Align* in the prompt menu.

The command bar reads: Enter an option [Angled/Horizontal alignment/Vertical alignment/Rotate view/Undo]:
3. Do one of the following:
 - To align two viewports horizontally:
 - Choose *Horizontal alignment* in the context menu or type *H* in the command bar, then press Enter.
 - The command bar reads: Specify basepoint:
 - Pick a point in the reference viewport.
The command bar reads: Specify point in viewport to be panned:
 - Pick a point in the viewport to be aligned.
The display in the viewport is moved vertically to align the two points horizontally.
 - To align two viewports vertically:

- Choose *Vertical alignment* in the context menu or type *V* in the command bar, then press Enter.
The command bar reads: Specify basepoint:
- Pick a point in the reference viewport.
The command bar reads: Specify point in viewport to be panned:
- Pick a point in the viewport to be aligned.
The display in the viewport is moved horizontally to align the two points vertically.

To align model space and paper space points

Command: ALIGNSPACE

Aligning 1 point

1. Type *alignspace* in the command bar, then press Enter.
The command bar reads: First alignment point in Model space:
2. Click inside a layout viewport, then pick a point.
The command bar reads: Second alignment point in Model space or <Return> for none:
3. Right click or press Enter.
The command bar reads: Alignment point in Paper space:
4. Pick a point in the layout.
The command bar reads: Activate the viewport which will be used for alignment.
5. Click inside the viewport you want to align.
6. Right click or press Enter to confirm.
The display in the selected viewport is moved to make the two points coincident.

Aligning 2 points

1. Type *alignspace* in the command bar, then press Enter.
The command bar reads: First alignment point in Model space:
2. Click inside a layout viewport, then pick a point.
The command bar reads: Second alignment point in Model space or <Return> for none:
3. Pick a second point in the layout viewport.
The command bar reads: First alignment point in Paper space:
4. Pick a point in the layout.
The command bar reads: Second alignment point in Paper space:
5. Pick a second point in the layout.
The command bar reads: Activate the viewport which will be used for alignment.
6. Click inside the viewport you want to align.
7. Right click or press Enter to confirm.
The display in the selected viewport is moved, rotated and/or scaled to make the points two by two coincident.

Layouts

Command: LAYOUT


In BricsCAD, you can create up to 255 layouts in a single drawing. Each layout represents a sheet of paper. For each layout you can specify the print area, print scale, lineweight scale, pen mappings, and add viewports, dimensions, a title block, and other geometry specific to the layout. The entities you add to a layout in paper space do not appear in model space.

Each layout requires at least one layout viewport. If the CREATEVIEWPORTS system variable is ON a viewport is added automatically in a new layout. If this variable is OFF, newly created viewports are empty. To add new layouts you can either define them yourself or you can import layouts from another drawing: template file (.dwt), drawing file (.dwg) or drawing interchange file (.dxf).

Viewports can display all or part of the drawing's model space entities.

The CACHELAYOUT system variable controls whether the content of layouts is cached. The extra memory required for caching is made up for by improvements to the memory requirements for displaying entities, the net result being that the memory needed to display drawings has been reduced. As a result of this switching between layouts is up to 10 times faster now.

Creating a new layout

1. Do one of the following:
 - Click the *New Layout* tool button () on the *Layout* or the *Insert* toolbar.
 - Choose *New Layout* in the *Insert* menu.
 - Type *layout* in the command bar, press Enter, then type *N* and press Enter.

The command bar reads: Enter a layout name. <Layout1>:

2. Do one of the following:
 - Type a unique name for the layout, then press Enter.
The name can be up to 255 characters in length and can contain letters, numbers, the dollar sign (\$), hyphen (-), and underscore (_), or any combination.
 - Right click or press Enter to accept the default name (e.g. Layout1).

The new layout tab is added.

3. Create at least one viewport (see Creating viewports in a layout).

If the CREATEVIEWPORTS system variable is ON a viewport is added automatically in a new layout.

Copying layouts

1. Right click the layout tab you want to copy.
A context menu displays.
2. Choose *Copy* in the context menu.
The command bar reads: Enter new layout name:
3. Type a new unique name in the command bar, then press Enter.
A copy of the selected layout is added.

Importing layouts

1. Right click the *Model* tab or one of the layout tabs.
A context menu displays.

2. Choose *From Template...* in the context menu.
The *Select Template From File* window opens.
3. Browse to the drawing that you want to import layouts from.
4. Click the *Open* button on the *Select Template From File* window.
The *Insert Layout(s)* window opens.
5. Select the layout(s) you want to import.
Press and hold the Ctrl key to select multiple layouts.
6. Click the OK button on the *Insert Layout(s)* window.
The selected layouts are imported.

Renaming layouts

1. Right click the layout tab you want to rename.
A context menu displays.
2. Choose *Rename* in the context menu.
The *Rename Layout* dialog opens.
3. Type a new unique name in the *Name* field of the *Rename Layout* dialog.
4. Click the OK button on the *Rename Layout* dialog.

Arranging the layout tabs

Drag the layout tab to the desired position

or

1. Right click the layout tab you want to move.
A context menu displays.
2. Do one of the following:
 - Choose *Move Right*.
 - Choose *Move Left*.
3. To move the selected layout tab do one of the following:
 - Select the number of tab positions.
 - Choose *Move to Last Layout* or *Move to First Layout*.

Deleting a layout

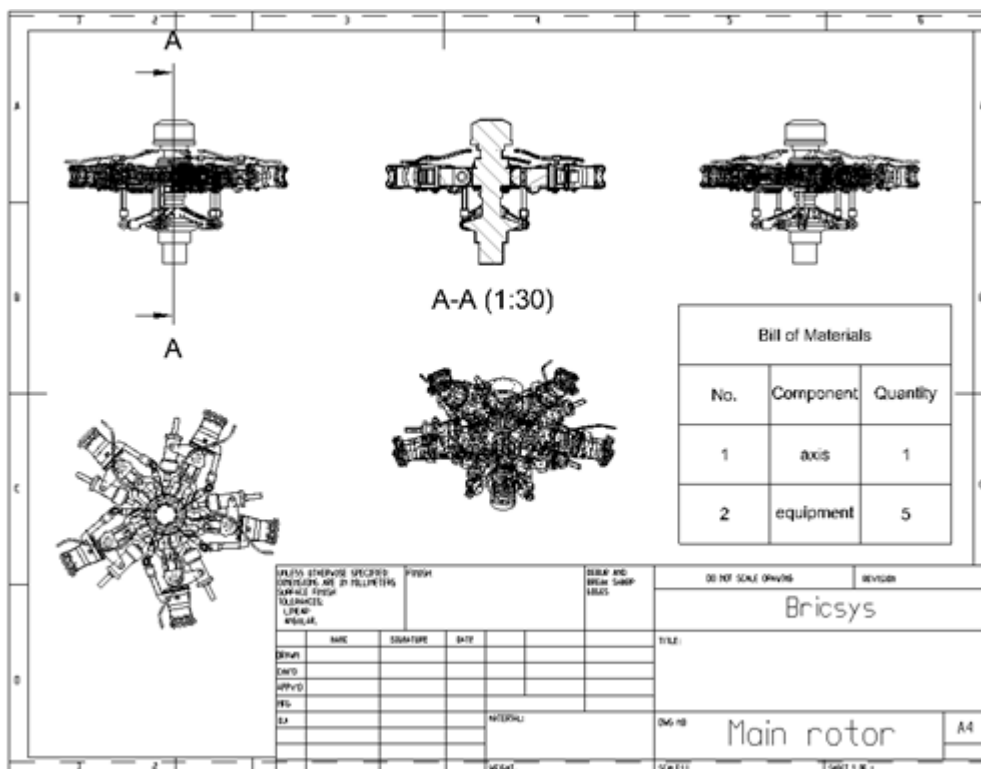
1. Right click the layout tab you want to move.
A context menu displays.
2. Choose *Delete* in the context menu.
The selected layout is deleted.

Generated Drawing Views

Commands: ViewBase, ViewSection, ViewUpdate and ViewExport

The Generated Drawing Views functionality allows to automatically generate associative orthographic and standard isometric views of a 3D solid model. All drawing views are placed in a paper space layout, they are not visible in model space.

Analytical hidden line removal (HLR) procedures are used to create the drawing views using standard 2D entities, mostly lines and arcs. Much like the result of the FlatShot command. Drawing views created in V14.1 will be automatically converted into HLR geometry by the ViewUpdate command.



The *ViewBase* command generates associative orthographic and standard isometric views of a 3D solid model in a paper space layout.

The *ViewSection* command creates a cross section view based on a standard drawing view generated by the *ViewBase* command in a paper space layout.

The *ViewUpdate* command Updates a selection of drawing views obtained by *ViewBase* and *ViewSection* when VIEWUPDATEAUTO = 0.

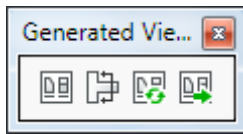
The *ViewExport* command exports the content of drawing views obtained by *ViewBase* and *ViewSection* to the Model Space of the drawing. This command can be used in paper space only.

NOTE Drawing views created in V14.2 will show correctly in BricsCAD V13. The most recent V13.2 version is recommended.

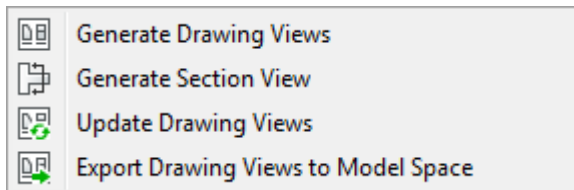
V14.2 drawing views do not display in V12 and older BricsCAD versions.

Commands and Toolbars

Tools to generate drawing views of your 3D solid model are available on the *Generated Views* toolbar, which is a flyout of the *View* toolbar:



And the *View* | *Generated Views* menu:



Standard Drawing Views

Standard views include multiview orthographic and isometric projections.

In a technical drawing, a multiview orthographic projection is an illustration technique in which up to six images of an object are generated, with each projection plane parallel to one of the coordinate axes of the object.

Isometric projection is a method represent three-dimensional objects in two dimensions in technical and engineering drawings. It is an axonometric projection in which the three coordinate axes appear equally foreshortened and the angles between any two of them are 120 degrees.

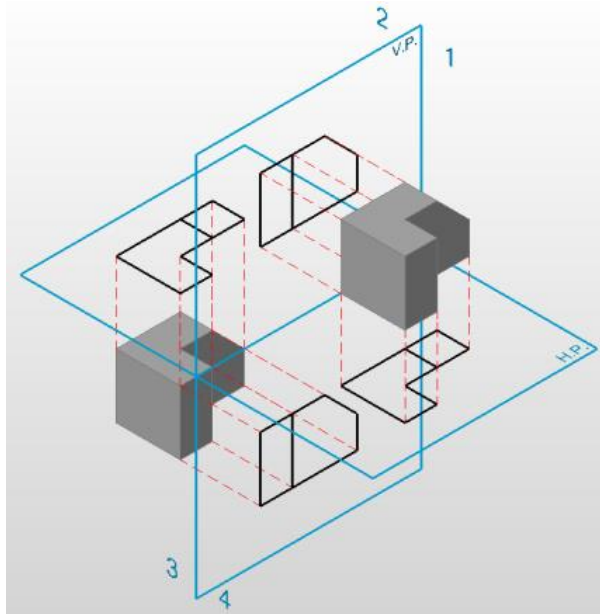
To generate the standard views of your 3D solid model, run the *ViewBase* command from model space.

You can choose between first angle (European) projection or third angle (American) projection.

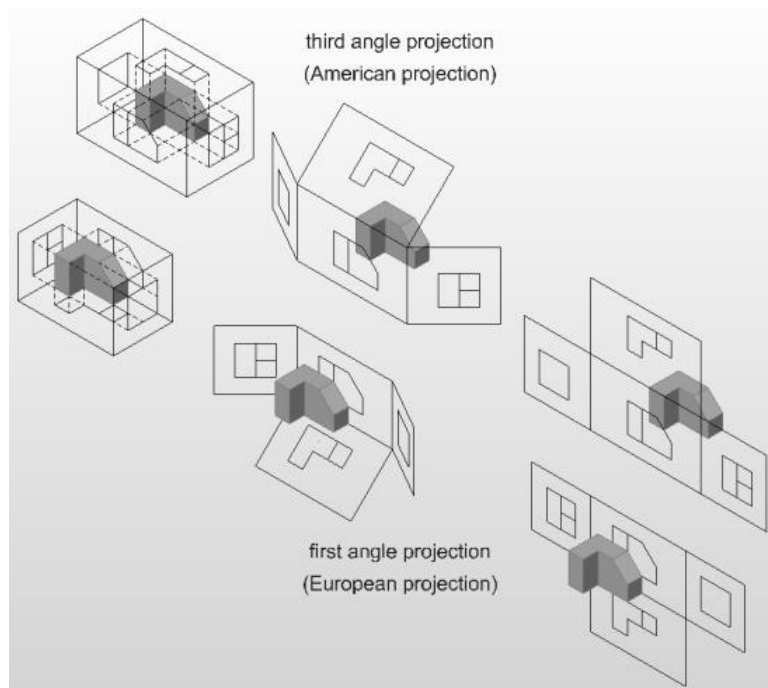
Orthographic projection (or orthogonal projection) is a means of representing a three-dimensional object in two dimensions. It is a parallel projection, where all the projection lines are orthogonal to the projection plane, resulting in every plane of the scene appearing in affine transformation on the viewing surface. It is further divided into multiview orthographic projections and axonometric projections.

Two projection planes, one horizontal (H.P.) and one vertical (V.P.) divide the space in four quarters.

- In the first angle projection system the model is placed in the first quarter space (first angle).
- In the third angle projection system the model is placed in the third quarter (third angle).



With multiview orthographic projections, up to six pictures of a 3D model are produced, with each projection plane parallel to one of the coordinate axes of the model. The views are positioned relative to each other according to either of two schemes: first-angle or third-angle projection. In each, the appearances of views may be thought of as being projected onto planes that form a 6-sided box around the model.



The projection type is set by the *Projection Type* option of the ViewBase command or by editing the PROJECTIONTYPE system variable:

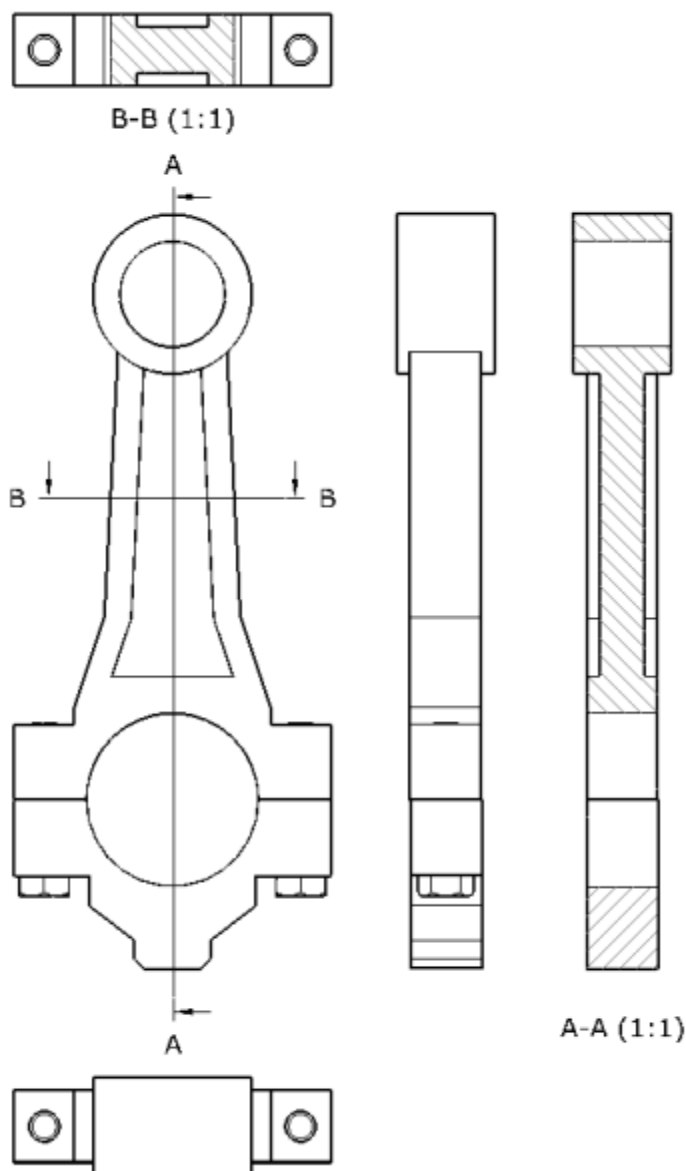
- 0 = First angle projection
- 1 = Third angle projection

The value of the PROJECTIONTYPE system variable is saved in the drawing.

Generating Cross Section Views

A cross section is the intersection of a 3D solid model with a section plane. In technical drawings, the internal parts of the 3D model are hatched in cross sections.

To generate section views of a 3D solid model, run the ViewSection command from the paper space layout where the standard drawing views exist.



Updating Drawing Views

By default BricsCAD checks whether the source 3D solid model was modified and automatically recalculates the drawing when:

- opening a paper space layout containing out-of-date views if the VIEWUPDATEAUTO system variable is ON.
- running the ViewUpdate command.
- running the BmUpdate command (BricsCAD Platinum only).

NOTE Hidden line removal calculation might take some time for complex 3D models. In such cases it is recommended to set VIEWUPDATEAUTO = OFF. The viewport border of out-of-date drawing views turns red to indicate an update is needed. When executing the ViewUpdate command, BricsCAD prompts you to either select the drawing views to be updated or update all drawing views.

Exporting Views to Model Space

The associativity mechanism of drawing views in a paper space layout allow limited control by the user. Such views can be moved or scaled, but you cannot edit the geometry. In order to get the full control to the drawing views geometry the ViewExport allow to move or copy drawing views to model space. Exported drawing views loose their associativity with the 3D model and become standard blocks, which can be exploded, edited, erased, etc&ldots;

Customizing Drawing Views

The first time the ViewBase command is launched, four new layers are created: two for basic orthographic views:

- *BM_Ortho_Hidden*: hidden lines in orthographic drawing views
- *BM_Ortho_Visible*: visible lines in orthographic drawing views
- *BM_Isometric_Hidden*: hidden lines in isometric drawing views
- *BM_Isometric_Visible*: visible lines in isometric drawing views

By default the display of the *BM_Isometric_Hidden* layer is switched off. Switch the layer on to have hidden lines in isometric views.

Edit the properties of these layers, such as linetype, lineweight or color, to modify the display of the drawing views.

Drawing Entities

In a drawing you can create a variety of different entity types. Drawing entities can be very simple such as lines, circles, arcs, points and rays or complex such as polylines, splines and planes. Apart from 2D entities you can also create 3D surface entities or 3D solids in BricsCAD.

To further complete your drawing BricsCAD provides a series of tools to add text, hatching and dimensions.

To create an entity you can choose between selecting the command in the Draw menu, using the tools on the Draw, Draw 3D and Solids toolbars or typing the command in the command bar.

When you use a tool or a drawing command, the program prompts you to enter coordinate points, such as endpoints or insertion points. You can enter the points or distances either using a mouse or by typing coordinate values in the command bar. As you draw, BricsCAD also displays a context-sensitive prompt menu with appropriate additional options for the type of entity you are drawing.

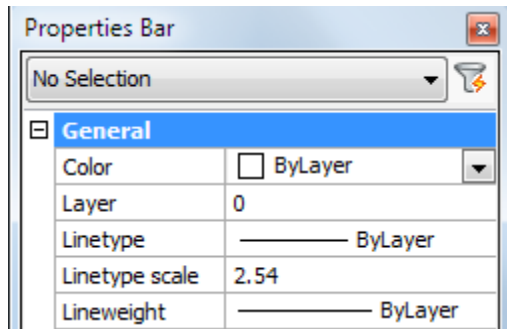
New entities are created on the current layer, using the current color, linetype and lineweight.

Overview

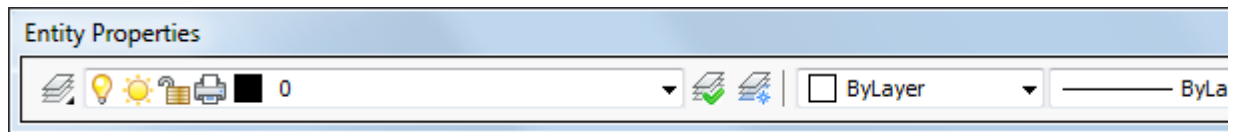
The following settings control the display and/or creation of entities.

Name	Command or System Variable	Description
Fill mode	FILL	Specifies whether multilines, traces, solids, hatches (including solid-fill) and wide polylines are filled in.
Current Entity Color	COLOR	Sets the color of new entities.
Whip Arcs	WHIPARC	Determines whether circles, arcs, ellipses, elliptical arcs and arc segments of polylines display as true curves or as a series of vectors.
Lineweight Display	LWDISPLAY	Controls whether lineweights display on the screen.
Default Lineweight	LWDEFAULT	Defines the default lineweight.
Lineweight Units	LWUNITS	Defines whether lineweights are expressed in millimeters or inches.
Current Entity Lineweight	CELWEIGHT	Sets the lineweight for new entities.
Current Entity Linetype Scale	CELTSCALE	Sets the linetype scaling factor for new entities.
Linetype Scale	LTSCALE	Sets the linetype scaling factor for all entities in the drawing (global linetype scale).
Current Entity Linetype	CELTYPE	Sets the linetype for new entities.
Current Layer	CLAYER	Sets the layer for new entities.

The current properties display in the *Properties Bar* and in the *Entity Properties* toolbar.



Properties Bar



Entity Properties toolbar





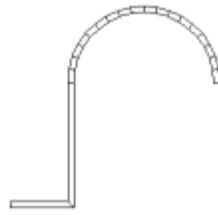
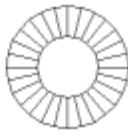
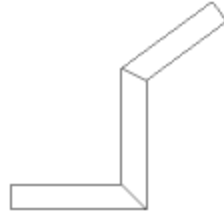
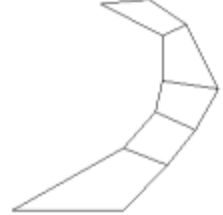
Fill Mode

Command: FILL

The *Fill* command toggles the display of filled and hatched areas. Specifies whether multilines, traces, solids, hatches (including solid-fill) and wide polylines are filled in.


If *FILLMODE* is OFF, all filled entities display and print as outlines.

You can reduce the time it takes to display or print a drawing by turning FILLMODE off.

	Polyline	Donut	Trace	Plane (Solid)
Fill On				
Fill Off				

Setting Fill Mode

1. To toggle *Fill Mode* on / off do one of the following:
 - Choose *Fill* in the *Settings* Menu.
 - Type *fill* or *fillmode* in the command bar, then type *T* + Enter.
2. Regenerate the viewport.

NOTE If multiple viewports are open, click the Regen All tool button () on the *View* toolbar (*Redraw/Regen* flyout) to regenerate all open viewports simultaneously.

Entity Color

Commands: COLOR and -COLOR

The *Color* command specifies the current color, through a dialog box.

The *-Color* command sets the current color, through the command bar.

An entity's color determines how it is displayed and how it prints. Entities are created in the current color.

You can choose between 255 *index colors* or define a *true color*.

Index Colors

Index color is the specification of the color of a pixel on a display screen using a an 8-bit color value, allowing up to 256 possible colors.

Each of the Index Colors has a unique number from 1 to 255. Seven of the index colors can also be referred to by name: red (1), yellow (2), green (3), cyan (4), blue (5), magenta (6) and white/black (7). Index color 7 displays white on a black screen background and black on a white screen background. Index color 7 always prints in black.

The two additional color properties are BYLAYER and BYBLOCK. These color properties cause an entity to adopt the color either of the layer or of the block in which it is a member. BYLAYER is color number 256, and BYBLOCK is color number 0. In all commands where you would use a color, you can indicate BYLAYER and BYBLOCK as well as by numbers 256 and 0, respectively.

Color BYLAYER:

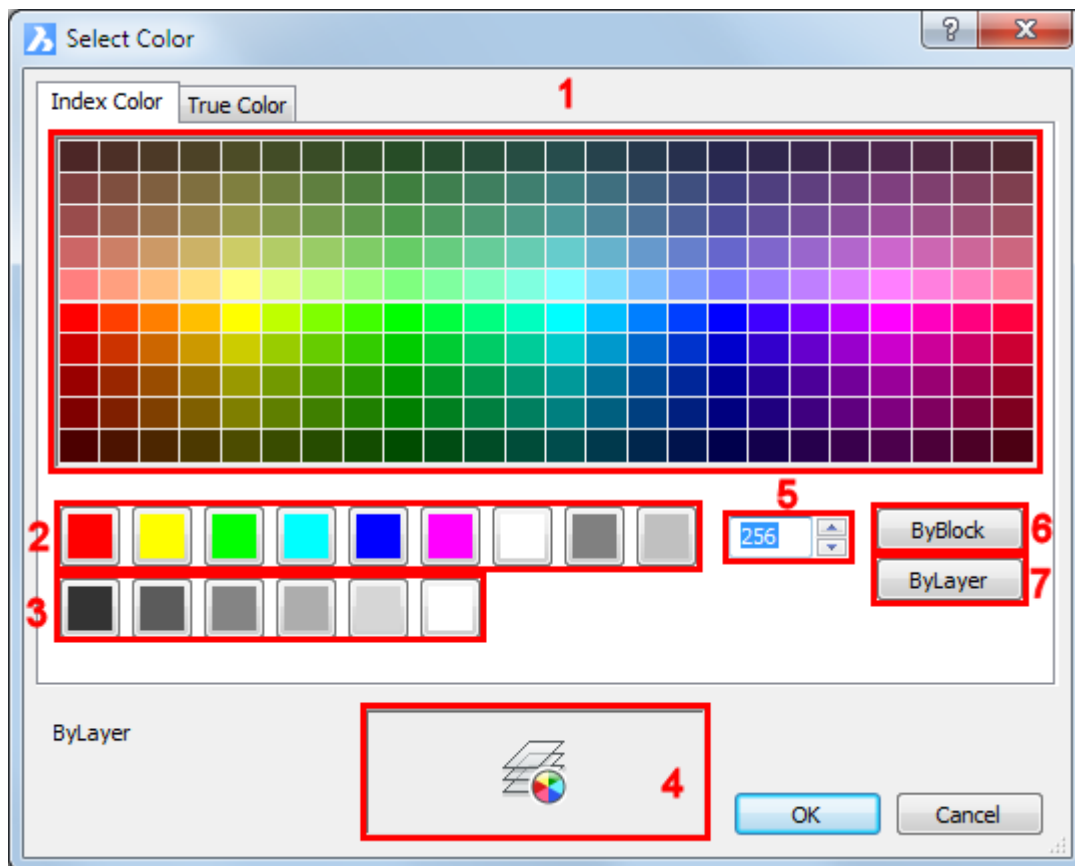
Entities which have a color *BYLAYER* adopt the color of their layer. This allows you to change the color of all such entities by adjusting the color of the layer.

Color BYBLOCK:

Entities which have a color *BYBLOCK* are drawn in index color 7 (black or white, depending on the screen background color). When included in a block definition, such entities adopt the color of the block.

NOTE Whether entities using an index color will print in this color, depends on the *Color Table* (CTB) or *Style Table* (STB) that is used for printing. Only if the *Color* setting in the CTB or STB definition file is set to *Use Object Color*, the printed color matches the entity color.

Index color dialog



1. Colors 10 - 249
2. Colors 1 to 9
3. Colors 250 - 255
4. Current Color
5. Color Number field
6. 'Color By Block' button
7. 'Color By Layer' button

True Colors

True color is the specification of the color of a pixel on a display screen using a 24-bit value, which allows up to 16 777 216 possible colors.

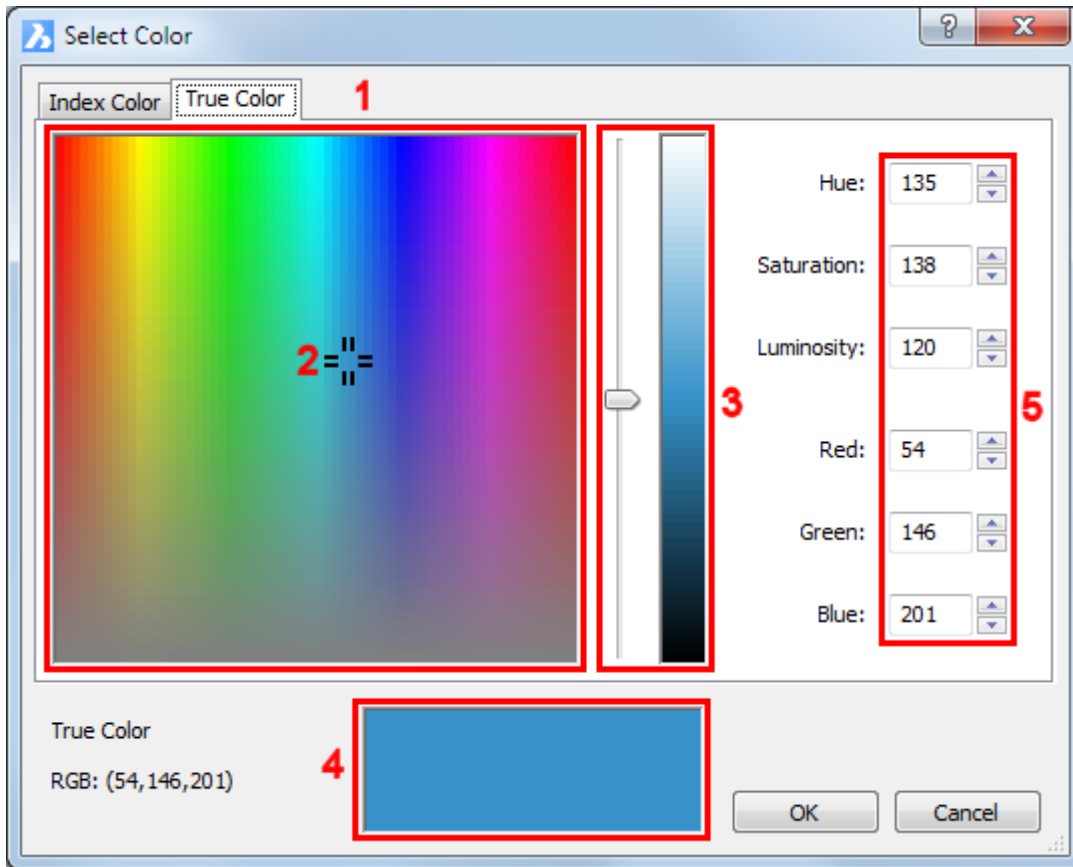
The number of bits used to define a pixel's color shade is its *bit-depth*. True color is sometimes known as *24-bit color*. Some new color display systems offer a 32-bit color mode. The extra byte, called the *alpha channel*, is used for control and special effects information.

True colors use a *RGB* color definition (Red, Green, Blue). Each of these parameters has a range from 0 to 255. The RGB definition for black is (0,0,0), the RGB definition for white is (255,255,255).

NOTE

- In drawings that use *Style Tables* (STB) to set up plotting configurations, entities in a true color will only be printed in this color if the *Color* setting in the STB definition file is set to *Use Object Color*.
- In drawings that use *Color Tables* (CTB) to set up plotting configurations, entities in a true color always print in this color.

True color dialog



1. Color Selection pane
2. Color Picker
3. Luminosity slider
4. Current Color
5. Color Parameter fields

Setting the current entity color

1. To open the *Select Color* dialog, do one of the following:
 - In the *Color* field of the *Entity Properties* toolbar, choose *Select Color*.
 - Click the *Select Color* tool button (🎨) on the *Settings* toolbar.
 - In the BricsCAD Properties Bar click *Color*, then choose *Select Color* in the drop down list.
 - Double click the *Color Field* in the Status Bar.
 - Type *color* in the command bar, then press Enter.
2. (option) To select an index color, click the *Index Color* tab on the *Select Color* dialog, then do one of the following:
 - Click one of the colored tiles.
 - Type the color number in the *Color Number* field.
3. (option) To define a true color, click the *True Color* tab on the *Select Color* dialog, then do one of the following:
 - Click the in *Color Selection* pane.

- Set the color parameters in the *Color Parameter* fields.
4. Click the *OK* button to confirm.

Lineweight

Lineweights determine how thick or thin entities appear on the screen and / or when printed. The following lineweights are available: BYLAYER, BYBLOCK, DEFAULT and many additional lineweights in millimeters or inches, depending on the *Lineweight Units* (LWUNITS) system variable..

You cannot assign lineweights to planes, points, TrueType fonts, and raster images.

New entities are drawn using the current lineweight.

Lineweight BYLAYER:

When you create an entity, it is created using the current lineweight. By default, the current lineweight for a new entity is BYLAYER. This means that the entity lineweight is determined by the current layer. When you assign BYLAYER, changing a layer's lineweight changes the lineweight of all such entities on that layer.

Lineweight BYBLOCK:

Entities created using lineweight BYBLOCK, are drawn using the DEFAULT lineweight until include them into a block. The entities then inherit the block's lineweight setting when you insert the block into a drawing.

DEFAULT lineweight:

The default lineweight is saved in the *Default Lineweight* (LWDEFAULT) system variable. Entities created using the default lineweight will be adjusted if the *Default Lineweight* setting is redefined.

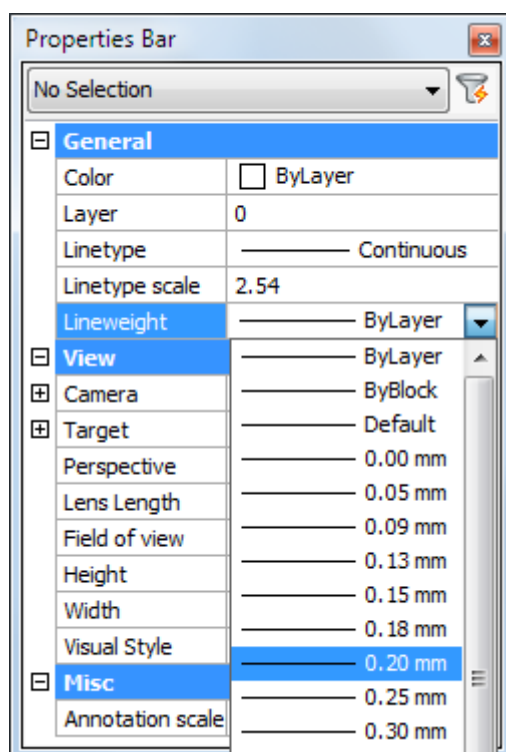
NOTE For entities using an index color, lineweights will be used for printing only if the *Color Table* (CTB) or *Plot Style* (STB) is defined as 'Use object lineweight'. Otherwise the line weight is defined by the objects color or plot style.

Entities which are created in a true color always use the object line weight when printed.

Setting the current lineweight

1. (option) On the BricsCAD Properties Bar, select *Lineweight*.
Make sure, no entity is selected.

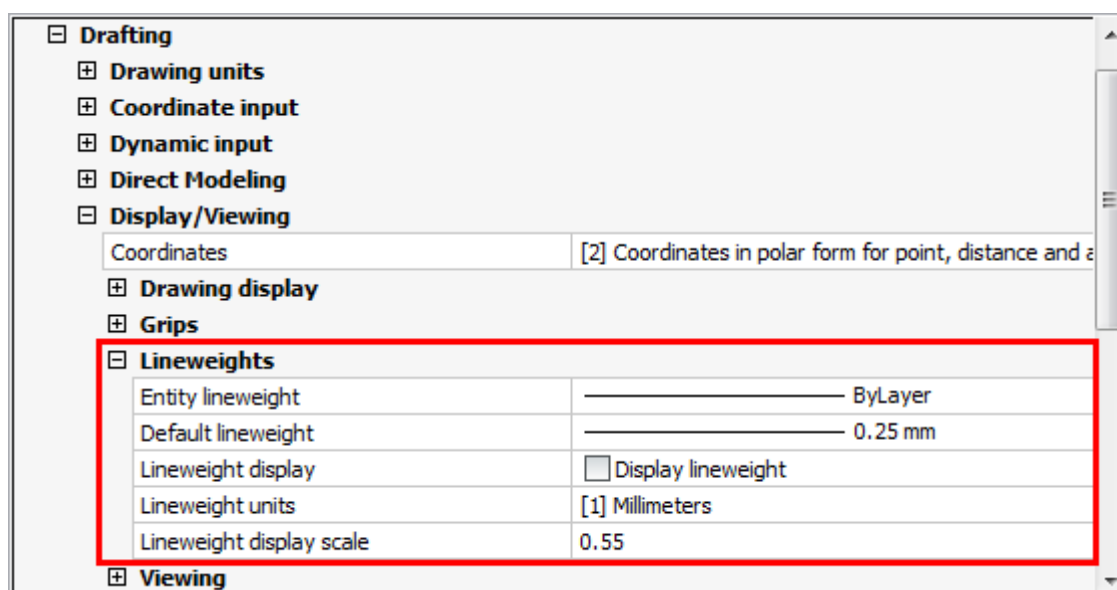
Choose a lineweight in the drop down list.



2. (option) On the *Entity Properties* toolbar, click the down arrow next to the *Lineweight* field, then choose a lineweight in the drop down list.

Defining the lineweight settings

1. Choose *Settings* in the *Settings* menu.
The Settings dialog opens.
2. Select the *Drawing settings* category.
3. Expand the *Display/Viewing* settings sub-category.
4. Expand the *Lineweights* settings group.



5. (option) Select the *Default Lineweight* (LWDEFAULT) system variable, then select a lineweight in the drop down list.
6. (option) Select the *Lineweight Display* (LWDISPLAY) system variable, then click the checkbox to toggle the display of lineweights.
7. (option) Select the *Lineweight Units* (LWUNITS) system variable, then choose *Millimeters* or *Inches*.
8. Close the *Settings* dialog.

NOTE Click the lineweight field (LWT) in the Status Bar to toggle the display of lineweights on/off.

Entity Linetype

You use different linetypes to differentiate the purpose of one line from another. A linetype consists of a repeating pattern of dots, dashes, or blank spaces. Lintypes determine the appearance of entities both on the screen and when printed. By default, every drawing has at least three lintypes: CONTINUOUS, BYLAYER and BYBLOCK. Your drawing may also contain an unlimited number of additional lintypes. Linetype definitions are saved in the drawing. New lintypes are either imported from other drawings or loaded from a linetype file (e.g. iso.lin or default.lin).

New entities are drawn using the current lineweight.

Linetype BYLAYER:

Entities which have a linetype BYLAYER adopt the linetype of their layer. This allows you to change the linetype of all such entities by adjusting the linetype of the layer.

Linetype BYBLOCK:

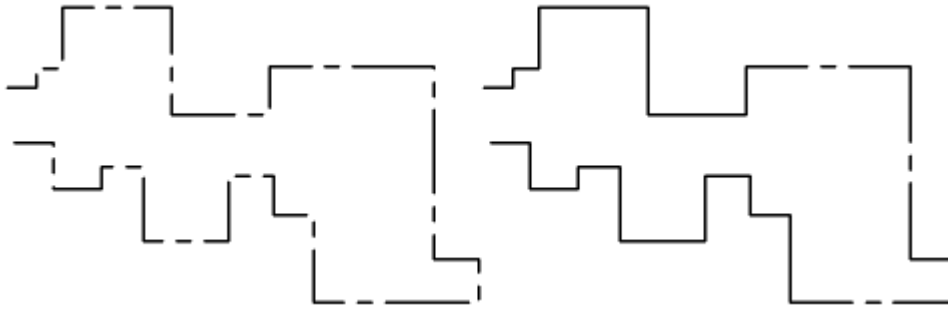
Entities created using linetype BYBLOCK, are drawn as continuous lines until you include them into a block. The entities then inherit the block's linetype setting when you insert the block into a drawing.

Linetype SCALE

Non-continuous lintypes consist of lines, gaps and dots put together in a variety of patterns. In complex lintypes also text or shapes can be included. In the linetype definition the length of the lines and gaps are defined in drawing units. To display the linetype correctly it needs to be scaled to match the dimensions of your drawing. The linetype scaling factor is defined by the *Linetype Scale* (LTSCALE) system variable. BricsCAD will always put a dash at the start and end of a line, polyline, arc or spline. The linetype pattern is also nicely centered, so that both ends of the entity look the same.

Because lintypes are affected by scale, paper space becomes a problem. A linetype scale that looks fine in model space is possibly going to look wrong in paper space. This problem is solved by the *Paperspace Linetype Scale* (PSLTSCALE) system variable. When set to 0 (zero) or OFF, lintypes are scaled the same in model space and in layouts, when set to 1 or ON, lintypes in layouts are drawn at the viewports scale.

Lintypes are normally generated from vertex to vertex. Polylines of which the vertices are very close together might be rendered as a continuous line, if the linetype pattern does not fit between two subsequent vertices. The *Polyline Generation* (PLINEGEN) system variable addresses this problem: when set to 1 or ON, the linetype is drawn from one end of the polyline to the other end, instead of from vertex to vertex.



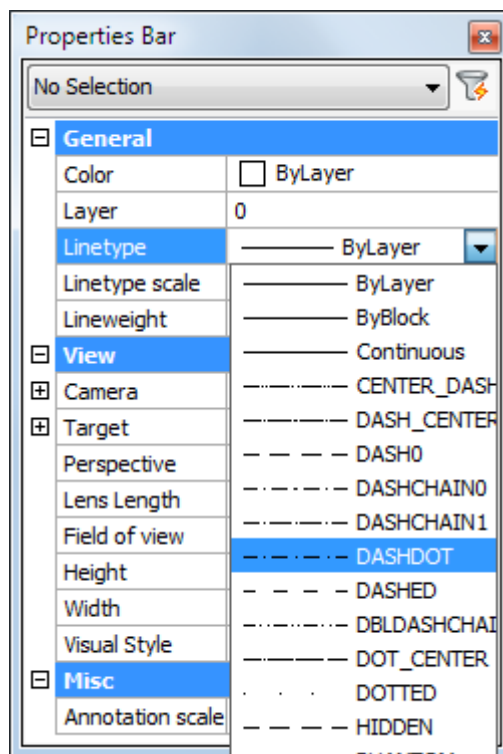
The same polyline drawn with *PLINEGEN* on (left) and off (right).

Setting the current linetype

Do one of the following:

- On the BricsCAD Properties Bar, select *Linetype*. Make sure, no entity is selected.

Choose a linetype in the drop down list.

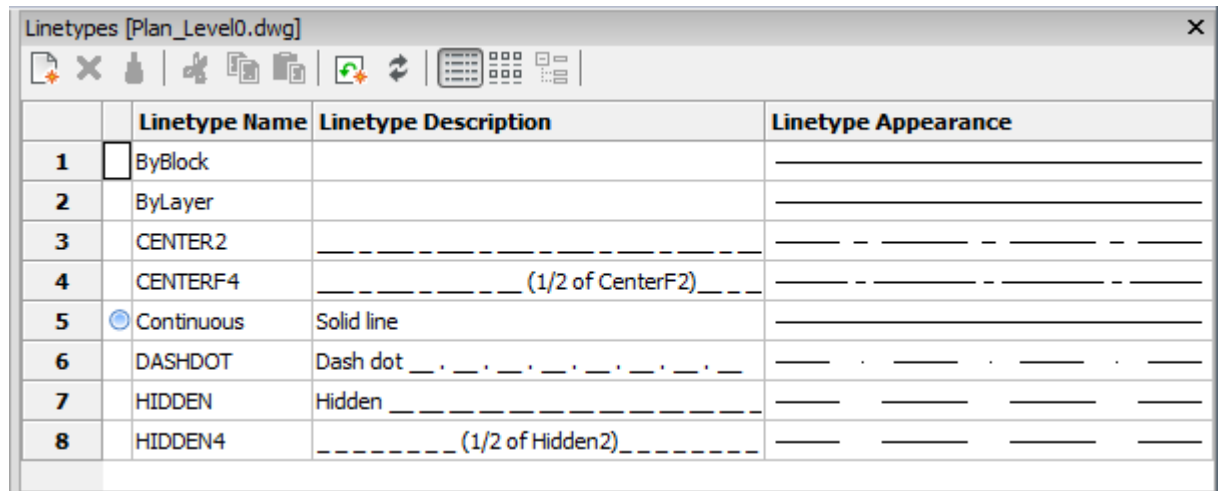


- On the *Entity Properties* toolbar, click the down arrow next to the *Linetype* field, then choose a linetype in the drop down list.
- Right click on the *Linetype* field in the *Status Bar*, then choose a linetype in the context menu.

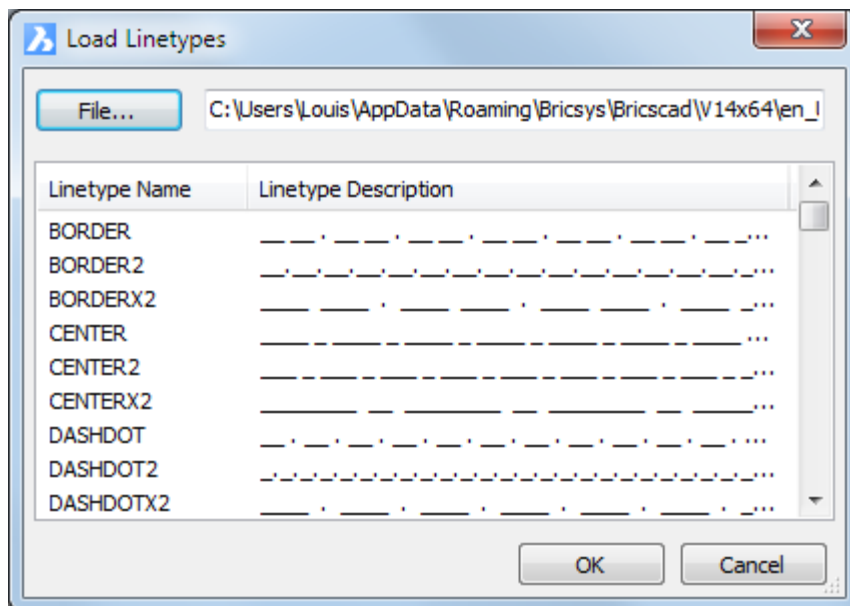
Adding a new linetype

1. In the Tools > Drawing Explorer choose *Linetype...*

The *Drawing Explorer - Linetypes* dialog displays.



2. Click the *New* tool button (📁) in the *Details* toolbar.
The *Linetypes* dialog opens.



3. (option) Click the *File* button on the *Linetypes* dialog to load a different linetype file (*.lin).
4. Select a linetype, then click the *OK* button.
The linetype is added.
5. (option) Click the linetype name to rename the linetype.
6. (option) click the linetype description to edit the description of the linetype.

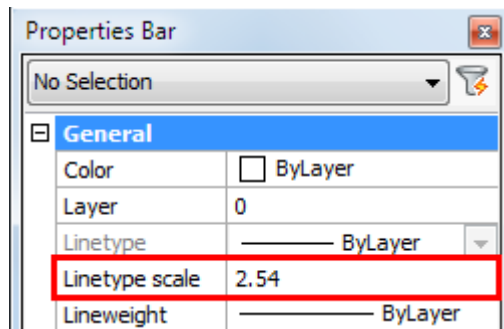
NOTE The MEASUREMENT system variable controls the which linetype file will be used to copy a new linetype definition to the drawing.

Default.lin is used if MEASUREMENT is OFF (Imperial), Iso.lin is used if MEASUREMENT is ON (Metric).

The linetype files (*.lin) sit in the *Support* folder of the Roamable root folder.

Setting the Entity Linetype Scale

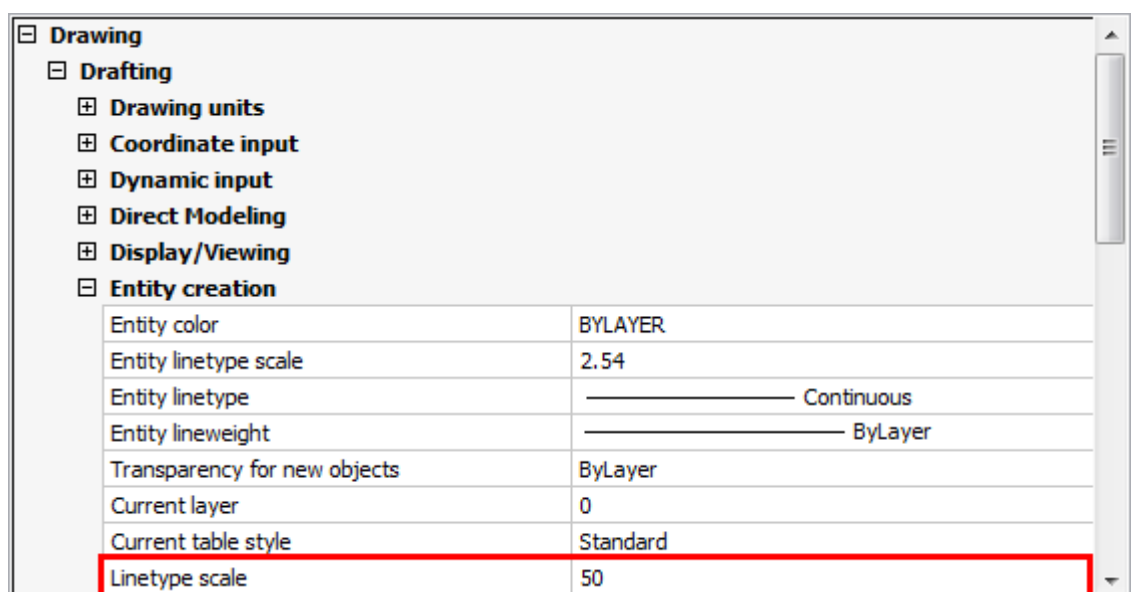
1. On the BricsCAD Properties Bar, select *Linetype scale*. Make sure, no entity is selected.
2. Type a new value in the Linetype scale field.



NOTE Linetype definitions in a *.lin file are expressed in drawing units: inches in default.lin, millimeters in iso.lin. If your drawing units are centimeters, set the linetype scale to 0.1 when using linetype definitions of iso.lin or 2.54 for linetype definitions of default.lin.

Setting the Global Linetype Scale

1. Choose *Settings* in the *Settings* menu. The Settings dialog opens.
2. Select the *Drawing settings* category.
3. Expand the *Entity Creation* settings sub-category.
4. Select the *Linetype Scale* setting (LTSCALE).



5. Type a new value in the *Global Linetype Scale* field, then press Enter. The drawing is regenerated.
6. Close the *Settings* dialog.

NOTE When printing Model Space: set the *Global Linetype Scale* with respect to the plot scale. E.g. if you want to plot your drawing at 1/50, set the LTSCALE setting to 50.

When printing a layout, do one of the following:

- If the *Paperspace Linetype Scale* setting (PSLTSCALE) is *Off* : set the *Global Linetype Scale* with respect to the plot scale. E.g. if you want to plot your drawing at 1/50, set the LTSCALE setting to 50.
- If the *Paperspace Linetype Scale* setting (PSLTSCALE) is *On*: set LTSCALE to 1.

Current Layer

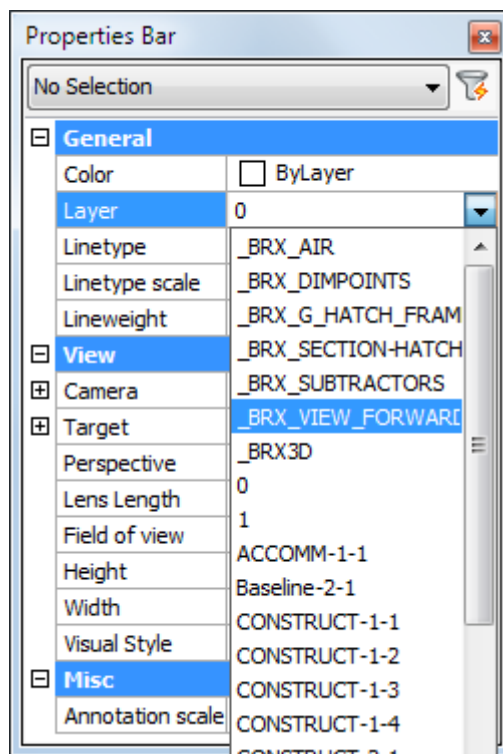
When you create new entities, they are drawn on the current layer. To draw new entities on a different layer, you must first make that layer the current layer.

Setting the current layer

Do one of the following:

- On the BricsCAD Properties Bar, select *Layer*. Make sure, no entity is selected.

Choose a layer in the drop down list.

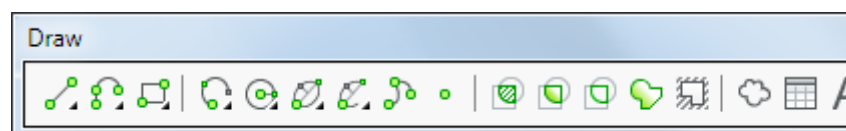


- On the Entity Properties toolbar, click the down arrow next to the Layer field, then choose a layer in the drop down list.
- Right click on the *Layer* field in the *Status Bar*, then choose a layer in the context menu.

Drawing 2D Overview

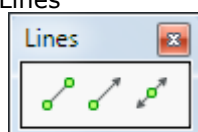
All tools to create 2D entities can be found either on the *Draw* toolbar or in the *Draw* menu.

Toolbars

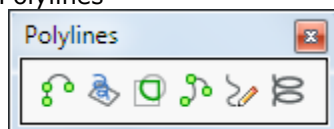


Draw flyouts:

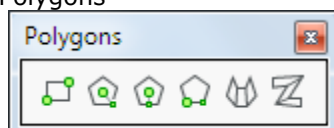
7. Lines



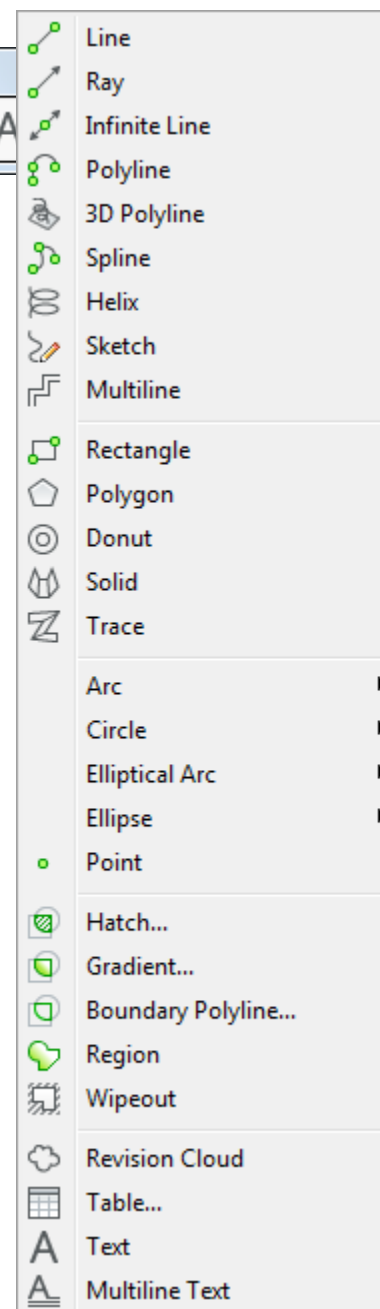
8. Polylines











9. Polygons



Menu



Icon	Tool Name	Command	Description
	Line	LINE	Creates single lines or a series of connected lines.
	Ray	RAY	Creates a line that starts at a point and extends to infinity.
	Infinite Line	XLINE	Creates a line through a given point, oriented at a specified angle and extending to infinity in both directions.
	Polyline	PLINE	Creates a single open or closed entity, composed of lines and/or arcs.
	3D Polyline	3DPOLY	Creates 3D polylines.
	Boundary Polyline	BOUNDARY	Calculates a closed polyline from a boundary set.
	Spline	SPLINE	Creates open or closed curved lines.
	Helix	HELIX	Creates a 2D spiral or a 3D helix.
	Sketch	SKETCH	Creates lines as if you were sketching, using a pencil.
	Rectangle	RECTANG	Creates Rectangles.
	Polygon	POLYGON	Creates equal sided polygons
	Solid	SOLID	Creates planes (solids).
	Trace	TRACE	Creates traces.
	Arc	ARC	Creates circular arcs.
	Circle	CIRCLE	Creates circles.
	Donut	DONUT	Creates donuts.
	Elliptical Arc	ELLIPSE+A	Creates elliptical arcs.
	Ellipse	ELLIPSE	Creates ellipses.
	Point	POINT	Creates points.
	Hatch	HATCH	Fills closed areas with repeating patterns.
	Gradient Fill	GRADIENT	Fills closed areas with solid fills in nine different patterns and one or two colors.
	Region	REGION	Creates regions from 2D entities.
	Wipeout	WIPEOUT	Creates wipeouts, blank areas in the drawing.
	Revision Cloud	REVCLOUD	Creates revision clouds.
	Table	TABLE	Creates a table entity in the drawing through a dialog box.
	Text	TEXT	Places single lines of text in the drawing.
	Multiline Text	MTEXT	Places multi-line text in the drawing through a dialog box.

Arcs


Command: ARC

The default method for drawing circular arcs is to specify three points: the start point, a second point and the endpoint.

Other methods to draw arcs can be found in the *Arc* submenu of the *Draw* menu and in the *Arcs* toolbar.

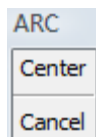
General procedure to draw an arc

1. Do one of the following.

- Click the *Arc* tool button () on the *Draw* toolbar.
- Type *arc* in the command bar, then press Enter.
- Type *A* in the command bar, then press Enter.

The command bar reads: Enter to use last point/Center/Follow/<Start of arc>:

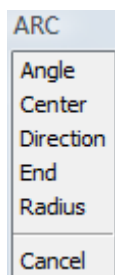
A prompt menu displays:



2. Specify the start point of the arc.

The command bar reads: Angle/Center/Direction/End/Radius/<Second point>:

The prompt menu changes:



3. Specify the second point of the arc.
The command bar reads: End point:
4. Specify the end point of the arc.

Editing an arc

1. Select the arc.
The properties of the arc display in the Properties bar.

[-] Geometry	
[-] Start point	38.5381, 191.1854, 0
X	38.5381
Y	191.1854
Z	0
[-] Center	3.7399, 153.1902, 0
X	3.7399
Y	153.1902
Z	0
[-] End point	-47.553, 158.0457, 0
X	-47.553
Y	158.0457
Z	0
Radius	51.5223
Start angle	48
End angle	175
Total angle	127
Arc length	114.2725
Area	1884.8676
[-] Normal	0, 0, 1
X	0
Y	0
Z	1

2. Type a new value in the *Center*, *Radius*, *Start angle* or *End angle*, then press Enter.
3. Press the Esc key to stop editing the arc.


Circles

Command: CIRCLE

The default method for drawing a circle is to specify a center point and radius.

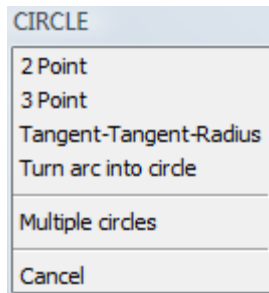
Other methods to draw circles can be found in the *Circle* submenu of the *Draw* menu and in the *Circles* toolbar.

General procedure to draw a circle

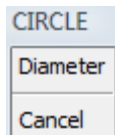
1. Do one of the following.
 - Click the *Circle* tool button () on the *Draw* toolbar.
 - Type *circle* in the command bar, then press Enter.
 - Type *C* in the command bar, then press Enter.

The command bar reads: 2Point/3Point/RadTanTan/Arc/Multiple/<Center of circle>:

A prompt menu displays:



- Specify the center point of the circle.
The command bar reads: Diameter/<Radius> <current radius>:
The prompt menu changes:



- Do one of the following:
 - Press Enter to accept the current radius.
 - Type a new radius and press Enter.
 - Click to define the radius graphically.

Editing a circle

- Select the circle.
The properties of the circle display in the Properties bar.

<input checked="" type="checkbox"/>	Geometry	
<input checked="" type="checkbox"/>	Center	85.0057, 163.4489, 0
	X	85.0057
	Y	163.4489
	Z	0
	Radius	59.6358
	Diameter	119.2716
	Circumference	374.7029
	Area	11172.8589
<input checked="" type="checkbox"/>	Normal	0, 0, 1
	X	0
	Y	0
	Z	1

- Type a new value in the *Center*, *Radius*, *Diameter*, *Circumference* or *Area* field, then press Enter.
- Press the Esc key to stop editing the circle.

Ellipses

Command: ELLIPSE

The default method for drawing an ellipse is to specify the endpoints of one axis of the ellipse and then specify a distance representing half the length of the second axis. The endpoints of the first axis determine the orientation of the ellipse. The longer axis of the ellipse is called the major axis, and the shorter one is the minor axis. The order in which you define the axes does not matter. The program determines the major and minor axes based on their relative lengths. Half the major axis is called the major radius, half the minor axis is the minor radius.

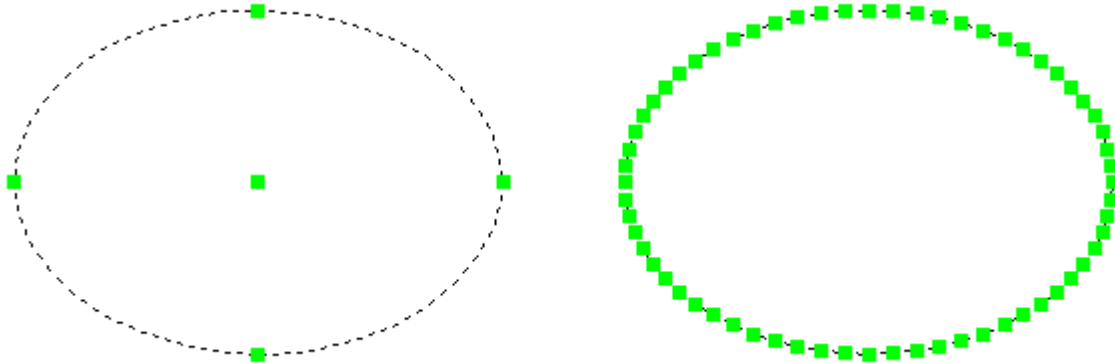
The ratio (minor radius divided by the major radius) defines the eccentricity of the ellipse. A circle is an ellipse with an eccentricity of 1.

Other methods to draw ellipses can be found in the *Ellipse* submenu of the *Draw* menu and in the *Ellipses* toolbar.

The *Polyline Ellipse* (PELLIPSE) settings variable: controls whether real ellipses or a polyline representation is created.

If the variable is ON, the *Ellipse* tool creates a closed polyline, if the variable is OFF real ellipses are created.

The difference between a real ellipse and a polyline representation of an ellipse is visible when you select the ellipse. A real ellipse has five handles: center point and the endpoints of the axes. A polyline representation of an ellipse is a closed polyline composed of arc segments.



real ellipse (left) and the polyline representation of an ellipse (right)

NOTE Since ellipses cannot be used as a clip boundary for external references or to create clipped paperspace viewports, you must use a polyline representation to do these jobs.

Setting the Polyline Ellipse (PELLIPSE) system variable


Do one of the following:

- In the command bar type *pellipse*, then press Enter.
- In the Settings dialog, go to *Drawing > Drafting > Entity Creation > Other entities*.

Other entities	
Circle radius	59.6358
Polyline ellipse	[0] Create real ellipses
Polygon sides	4
Shape name	

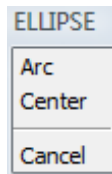
General procedure to draw an ellipse

1. Do one of the following.

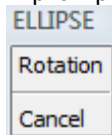
- Click the *Ellipse* tool button () on the *Draw* toolbar.
- Type *ellipse* in the command bar, then press Enter.
- Type *EL* in the command bar, then press Enter.

The command bar reads: Arc/Center/<First end of ellipse axis>:

A prompt menu displays:



2. Specify the the first end of the ellipse axis.
The command bar reads: Second end of axis:
3. Specify the second end of the ellipse axis.
The command bar reads: Rotations/<Other axis>:
A prompt menu displays:



4. Specify the length of half the other axis of the ellipse.

NOTE If the *Polyline Ellipse* variable is off, the *Arc* option is not available.

Editing an ellipse

1. Select the ellipse.
The properties of the ellipse display in the Properties bar.

☐	Geometry	
☐	Start point	-55.838, 242.3358, 0
	X	-55.838
	Y	242.3358
	Z	0
☐	Center	-17.4752, 242.3358, 0
	X	-17.4752
	Y	242.3358
	Z	0
☐	End point	-55.838, 242.3358, 0
	X	-55.838
	Y	242.3358
	Z	0
	Major radius	38.3628
	Minor radius	22.6197
	Radius ratio	0.5896
	Start angle	0
	End angle	0
☐	Major axis vector	-38.3628, 0, 0
	X	-38.3628
	Y	0
	Z	0
☐	Minor axis vector	0, -22.6197, 0
	X	0
	Y	-22.6197
	Z	0
	Area	2726.1383

2. Type a new value in the *Center*, *Major radius*, *Minor radius*, *Radius ratio*, *Start angle* or *End angle* field, then press Enter.
3. Press the Esc key to stop editing the ellipse.

Elliptical Arcs


Command: ELLIPSE + A

An elliptical arc is a portion of an ellipse. The default method for drawing an elliptical arc is to specify the endpoints of one axis of the ellipse, and then specify a distance representing half the length of the second axis. Then you specify the start and end angles for the arc, measured from the center of the ellipse in relation to its major axis.

Other methods to draw elliptical arcs can be found in the *Ellipse* submenu of the *Draw* menu and in the *Elliptical Arcs* toolbar.

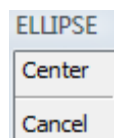
General procedure to draw an elliptical arc

1. Do one of the following.

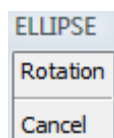
- Click the *Elliptical Arc* tool button () on the *Draw* toolbar.
- Launch the Ellipse command using the *Arc* option.

The command bar reads: Center/<First end of ellipse axis>:

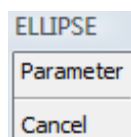
A prompt menu displays:



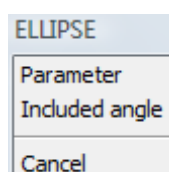
2. Specify the first end of the ellipse axis.
The command bar reads: Second end of axis:
3. Specify the second end of the ellipse axis.
The command bar reads: Rotation/<Other axis>
A prompt menu displays:



4. Specify the length of half the other axis of the ellipse.
The command bar reads: Parameter/<Start angle of arc>:
The prompt menu changes:



5. Specify the start angle of the elliptical arc.
The elliptical arc is drawn dynamically when you move the crosshairs.
The command bar reads: Parameter/Included/<End angle>:
The prompt menu changes:



- Specify the end angle of the elliptical arc.

Editing an elliptical arc

- Select the elliptical arc.
The properties of the elliptical arc display in the Properties bar.


☐	Geometry	
☐	Start point	-10.743, 100.3847, 0
	X	-10.743
	Y	100.3847
	Z	0
☐	Center	-25.58, 70.3336, 0
	X	-25.58
	Y	70.3336
	Z	0
☐	End point	17.1995, 57.1941, 0
	X	17.1995
	Y	57.1941
	Z	0
	Major radius	54.7481
	Minor radius	29.1773
	Radius ratio	0.5329
	Start angle	234
	End angle	154
☐	Major axis vector	-54.0321, -8.8252, 0
	X	-54.0321
	Y	-8.8252
	Z	0
☐	Minor axis vector	4.7033, -28.7958, 0
	X	4.7033
	Y	-28.7958
	Z	0
	Area	4196.6226

- Type a new value in the *Center*, *Major radius*, *Minor radius*, *Radius ratio*, *Start angle* or *End angle* field, then press Enter.
- Press the Esc key to stop editing the elliptical arc.

Freehand sketches

Command: SKETCH

A freehand sketch consists of a series of straight line segments, created either as individual line entities or as a polyline. Before you begin creating a freehand sketch, you must set the length, or increment, of each segment. The smaller the segments, the more accurate your sketch, but small segments increase the file size.

After you specify the length of the sketch segments, the crosshairs change to a pencil tool ().

A freehand sketch line is not added to the drawing until you "write" the sketch into your drawing.

Setting the Sketch Poly (SKPOLY) system variable

Whether a chain of individual lines or a polyline is created by the *Sketch* tool is controlled through the *Sketch Poly* (SKPOLY) system variable. When the setting is *ON* the *Sketch* tool creates polylines


Do one of the following:

- In the command bar type *skpoly*, then press Enter.
- In the Settings dialog, go to *Drawing > Drafting > Entity Creation > Freehand Sketches*.



Creating a freehand sketch

1. Do one of the following

- Click the *Sketch* tool button () on the *Draw* toolbar.
- Choose *Sketch* in the *Draw* menu.
- Type *sketch* in the command bar, then press Enter.

The command bar reads: Length of segments <current length>:

2. Do one of the following to define the length of the segments::

- Press Enter to accept the current length.
- Type a new value in the command bar.
- Click two points to define the length graphically.

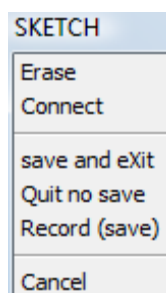
The length of the segments is saved to the *Sketch Increment* (SKETCHINC) system variable.

3. Click to start sketching (pen down). The crosshairs are replaced by the *Pencil* tool (




The command bar reads: Press ENTER or click to toggle Pen/Quit without save/Record (save)/Erase/Connect/Straight/eXit with save::

A prompt menu displays:



4. Move the cursor to sketch.
The sketched line displays in light green.
5. (option) Click to stop sketching (pen up), then click again (pen down) to resume.

6. (option) Choose *Erase* on in the prompt menu or type *E* + Enter in the to start erasing sketch lines.
The cursor changes to the *Erase* tool ().
 - Move the cursor to the end of a sketch line to start erasing.
 - Click to stop erasing and resume sketching.
7. (option) Choose *Record (save)* in the prompt menu or type *R* + Enter to save the sketch lines, and keep sketching.
8. (option) Choose *save and eXit* in the prompt menu or type *X* + Enter to save the sketch lines and exit the sketch command.
9. (option) Choose *Quit no save* in the prompt menu or type *Q* + Enter to exit the sketch command without saving.

Infinite Lines


Command: XLINE

The *Xline* command creates infinite lines.

An *Infinite Line* is a line through a given point, oriented at a specified angle and extending to infinity in both directions. Because infinite lines extend to infinity, they are not calculated as part of the drawing extents. The default method for drawing an infinite line is to select a point and then specify its direction.

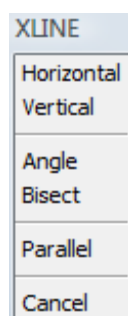
Rays and Infinite Lines or sometimes referred to as construction lines.

Drawing infinite lines

1. Do one of the following:
 - Click the *Infinite Line* tool button () on the *Lines* flyout of the *Draw* toolbar.
 - Choose *Infinite Line* in the *Draw* menu.
 - Type *xline* in the command bar, then press Enter.

The command bar reads: Xline: Bisect/Horizontal/Vertical/Angle/Parallel/<Point along line>:

A prompt box displays:



2. Specify the start point of the infinite line.
The infinite line is created.
A new infinite line through the same start point displays dynamically.
3. (option) Specify the direction of the infinite line.
The infinite line is created.
A new infinite line through the same start point displays dynamically.
4. (option) Repeat step 3 to draw a series of infinite lines, through the same start point.
5. Right click to stop drawing infinite lines.
6. (option) Right click to restart.


Lines

Command: LINE

A line consists of two points: a start point and an endpoint. Using the *Line* command you can draw a series of connected lines, but each line is considered a separate line entity.

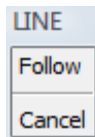
Drawing lines

1. Do one of the following

- Click the *Line* tool button () on the *Draw* toolbar.
- Choose *Line* in the *Draw* menu.
- Type *line* in the command bar, then press Enter.
- Type *L* in the command bar, then press Enter.

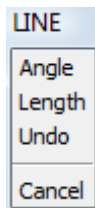
The command bar reads: ENTER to use last point/Follow/<Start of line>:

A prompt menu displays:



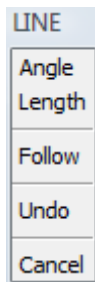
2. Specify the start point.
The line displays dynamically.
3. Specify the end point.
The line is drawn and the following line displays dynamically starting in the endpoint of the first line.
 - Using Ortho Mode or Polar Tracking you can type the length of the line in the command bar.
 - Using Snap Tracking you can position the end point with respect to the start point and one or two snap points on entities in the drawing.

The prompt menu changes:

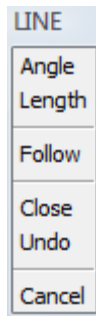


4. (option) Repeat step 3 to draw a series of connected lines.

When the second line is drawn, the options in the prompt menu are:



After the third line, the options in the prompt menu are:

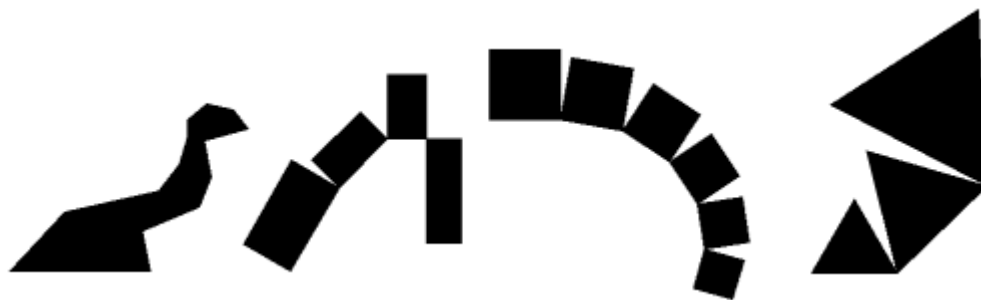


5. Right click to stop drawing lines.
6. (option) Right click to restart.

Solids

Command: SOLID

Solids are rectangular, square, triangular or quadrilateral areas filled with a solid color.



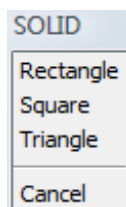
Left to right: quadrilateral, rectangular, square and triangular solids

Drawing solids

1. Do one of the following:
 - Click the *Solid* tool button (🔷) on the *Polygons* flyout of *Draw* toolbar.
 - Choose *Solid* in the *Draw* menu.
 - Type *solid* in the command bar, then press Enter.

The command bar reads: Rectangle/Square/Triangle/<First point of solid>:

A prompt menu displays:



2. Specify the first point of the solid.
3. Specify the second point of the solid.
4. Specify the third and fourth point of the solid.
The solid is created.
5. (option) Repeat step 4 to add more solids.
A series of connected solids is created.
6. Right click to conclude.

NOTE When Fill Mode is turned off, all filled entities, such as wide polylines and solids, display and print as outlines.

Points

Command: POINT

Since points are dimensionless, a single pixel should be the (nearly) correct representation on the screen. But this is hardly visible, especially in a complex drawing. Therefore you can choose between a number of possible display styles.

The point display style is chosen by means of the *Point Display Mode* (PDMODE) system variable. The size of the point representation style is controlled through the *Point Display Size* (PDSIZE) system variable.

Defining the point display settings

1. Open the Settings dialog.
2. In the *Drawing* settings category, expand the *Entity Creation* sub-category.
3. Expand the *Points* settings group, then expand the *Point display mode* setting.

Points	
Point display mode	0x0063
0x000F	x
0x0020	<input checked="" type="checkbox"/> circle
0x0040	<input checked="" type="checkbox"/> square
Point display size	0.0000

	<input type="checkbox"/> circle <input type="checkbox"/> square	<input checked="" type="checkbox"/> circle <input type="checkbox"/> square	<input type="checkbox"/> circle <input checked="" type="checkbox"/> square	<input checked="" type="checkbox"/> circle <input checked="" type="checkbox"/> square
none		○	□	◻
.	.	◦	◻	◻
+	+	⊕	⊕	⊕
x	×	⊗	⊗	⊗
°	°	⦿	⦿	⦿


Point display modes

1. Set the *Point Display Mode* setting:

- Choose the *point location marker* style: *point*, *none*, *X* or *|*
 - Choose to add a *circle* and/or a *square*.
2. Set the *Point Display Size* setting.
 - Relative: Type 0 (zero) to scale the point display style at 5% of the screen.
 - Absolute: Type the size of the point display style in drawing units.
 3. Close the *Settings* dialog.

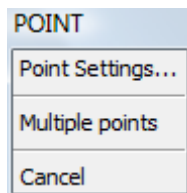
NOTE If you need to draw a 'dot', use a Donut with a zero inside diameter instead of a point entity.

Drawing points

1. Do one of the following
 - Click the *Point* tool button () on the *Draw* toolbar.
 - Choose *Point* in the *Draw* menu.
 - Type *point* in the command bar, then press Enter.

The command bar reads: Settings/Multiple/<Location of point>:

A prompt menu displays:



2. Specify the location of the point.
The point is created.




NOTE If the *Point Display Size* setting is set to 0 (zero), use the Regen tool to resize the display of the points to 5% of the screen size after zooming in or out.

Polygons

Command: POLYGON


Polygons are closed polylines comprised of a minimum of three and a maximum of 1,024 equal-length sides.

In BricsCAD you can draw polygons using the following methods:

- *Centre - Vertex* (): first define the center point, then the vertex (= radius of the circumscribed circle method).
- *Center - Side* (): first define the center point, then the midpoint of a side (= radius of the inscribed circle method).
- *Edge* (): define the length of the side of the polygon.

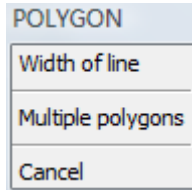
Drawing polygons

1. Do one of the following

- Click the *Polygon, Center - Vertex* tool button () on the *Polygons* flyout of *Draw* toolbar.
- Choose *Polygon* in the *Draw* menu.
- Type *polygon* in the command bar, then press Enter.

The command bar reads: Polygon: Multiple/Width of line/<Number of sides> <4>:

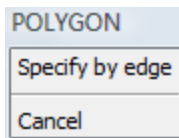
A prompt menu opens:



2. Specify the number of sides, then press Enter.

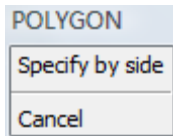
The command bar reads: Specify by: Edge/<Center of polygon>:

The prompt menu changes:



3. Specify the center of the polygon.

The command bar reads: Specify by: Side/<Select vertex point>:
the prompt menu changes:



4. Specify a vertex point of the polygon.
The polygon is created.

NOTE When Fill Mode is turned off, all filled entities, such as wide polylines and planes, display and print as outlines.

Polylines

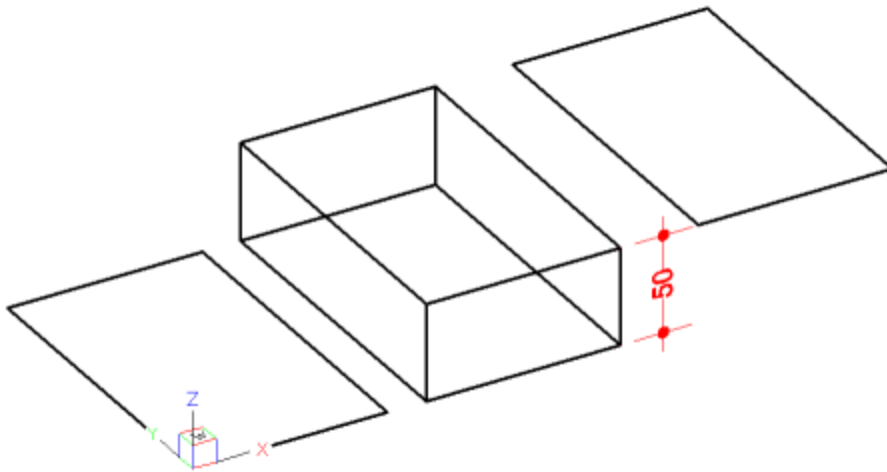
Commands: PLINE, BOUNDARY and -BOUNDARY

A *polyline* is an open or closed sequence of connected line and/or arc segments, which are treated as a single entity. Each segment of a polyline can have a width that is either constant or tapers over the length of the segment.

The *Pline* command interactively creates open and closed polylines entering points.

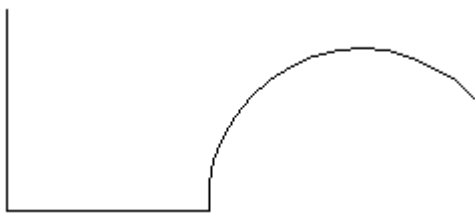
The *Boundary* and *- Boundary* commands create closed polylines from an enclosed area, defined by other entities.

The *THICKNESS* system variable specifies the height of a polyline. If *THICKNESS* = 0 (zero) there is no visual difference between polylines and a series of lines and/or arcs.

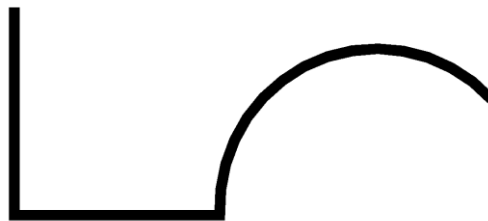


THICKNESS = 0 (left), THICKNESS = 50 (middle), ELEVATION = 50 (right)

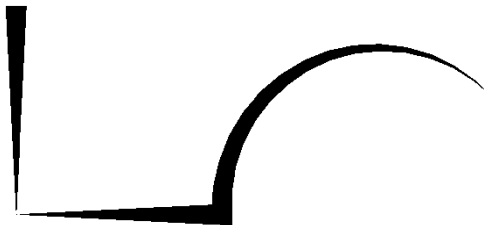
When a polyline is edited, you can modify the entire polyline or change individual segments and you can add or remove vertices.



zero width polyline



constant width polyline




tapered width polyline

NOTE When Fill Mode is turned OFF, all filled entities, such as wide polylines and planes, display and print as outlines.

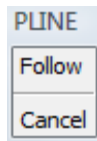
Drawing polylines

1. Do one of the following

- Click the *Polyline* tool button () on the *Draw* toolbar.
- Choose *Polyline* in the *Draw* menu.
- Type *pline* in the command bar, then press Enter.
- Type *PL* in the command bar, then press Enter.

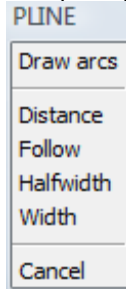
The command bar reads: ENTER to use last point/Follow/<Start of polyline>:

A prompt menu displays:



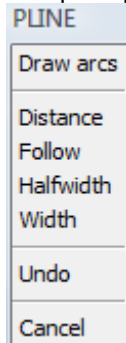
2. Specify the start point of the polyline.

The prompt menu changes:



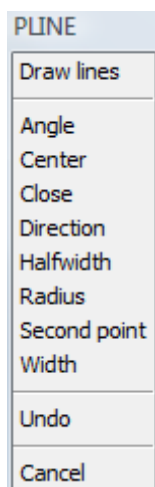
3. Specify the second point of the polyline.

The prompt menu changes:



4. (option) Repeat step 3 to add more straight segments.
5. (option) Do one of the following to start drawing arc segments:
 - Type *A*, then press Enter.
 - Choose *Draw arcs* in the prompt menu.

The prompt menu changes:




An arc segment displays dynamically. The arc is tangent to the previous line segment.

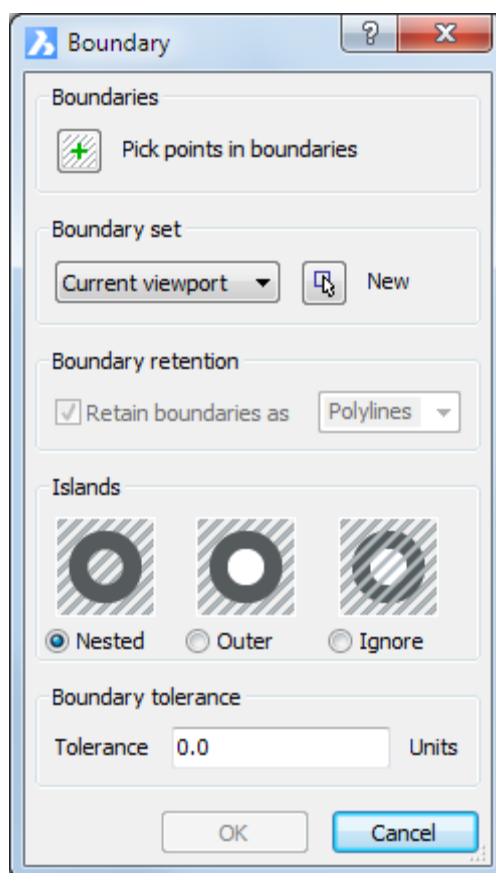
6. Specify the endpoint of the arc.
7. (option) Repeat step 6 to add more arc segments.


8. (option) Do one of the following to start drawing line segments:
 - Type *L*, then press Enter.
 - Choose *Draw lines* in the prompt menu.
9. Specify the endpoint of the line segment.
10. Right click or choose *Done* in the prompt menu to stop.
11. (option) Right click to restart.

Creating boundary polylines

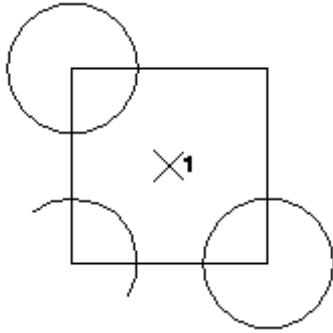
1. Do one of the following
 - Click the *Boundary Polyline* tool button () on the *Draw* toolbar.
 - Choose *Boundary Polyline* in the *Draw* menu.
 - Type *boundary* in the command bar, then press Enter.

The *Boundary* dialog box opens.

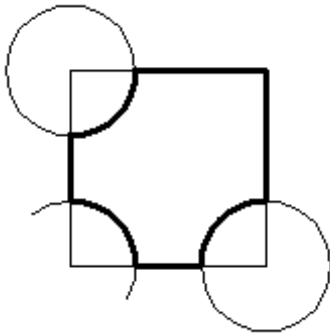


2. (option) Click the *Select* button ()
 The *Boundary* dialog closes.
 Select the boundary entities in the drawing, then Press Enter.
 The *Boundary* dialog opens.
3. Click the *Pick points in boundaries* button on the *Boundary* dialog box.
 The dialog box closes.


4. Click in the area where you want to create the boundary polyline: point 1 in the image below.



5. The boundary polyline displays in dashed lines.
6. (option) Keep picking points to create more boundary polylines.
7. Right click to accept the polyline(s).
The *Boundary* dialog box opens again.
8. Click the *OK* button on the *Boundary* dialog box.
The polyline(s) is (are) created.



NOTES

- If you select the boundary entities before launching the *Boundary Polyline* tool, you can make the selection set active by pressing the *Select* button () in step 2.
- If you type *-boundary* (don't forget the "minus" sign) in the command bar, you are prompted to click the area where you want to create the polyline(s) immediately (no dialog opens) then right click to create the polyline(s).

Rays

Command: RAY


A *Ray* is a line that starts at a point and extends to infinity. Because rays extend to infinity, they are not calculated as part of the drawing extents.

The default method for drawing a ray is to select the start point of the ray, and then specify its direction.

Rays and Infinite Lines or sometimes referred to as construction lines.

Drawing rays

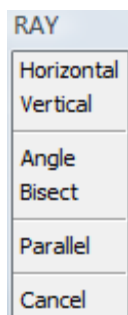
1. Do one of the following:

- Click the *Ray* tool button () on the *Lines* flyout of the *Draw* toolbar.
- Choose *Ray* in the *Draw* menu.
- Type *ray* in the command bar, then press Enter.

The command bar reads: Infinite ray:

Bisect/Horizontal/Vertical/Angle/Parallel/<Start of Ray>:

A prompt menu opens:



2. Specify the start point of the ray.
The ray displays dynamically.
3. Specify the direction of the ray.
The next ray displays dynamically.
4. (option) Keep specifying directions to draw a series of rays, starting from the same point.
5. Right click to stop drawing rays.

Rectangles

Command: RECTANG

A rectangle is created as a closed, four sided polyline.


A rectangle is drawn by specifying two opposite corners. The *Dimensions* option allows to specify the width and height of the rectangle.

Unless the *Rotated* option is chosen, the sides of a rectangle are always parallel to the x-axis and y-axis of the current coordinate system.

NOTE When Fill Mode is turned off, all filled entities, such as wide polylines and planes, display and print as outlines.

Drawing rectangles

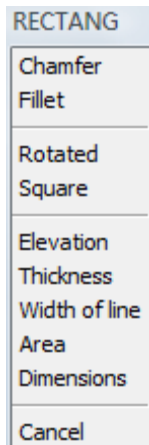
1. Do one of the following

- Click the *Rectangle* tool button () on the *Draw* toolbar.
- Choose *Rectangle* in the *Draw* menu.
- Type *rectang* in the command bar, then press Enter.
- Type *rec* in the command bar, then press Enter.

The command bar reads:

Chamfer/Elevation/Fillet/Rotated/Square/Thickness/Width/Area/Dimensions<select first corner of rectangle>:

A prompt menu opens:



- Specify the first corner of the rectangle.
The rectangle displays dynamically when you move the cursor.
- Specify the opposite corner of the rectangle.
The rectangle is created.
- (option) Right click to draw more rectangles.

NOTE Because a rectangle is created as a closed polyline, rectangles have a direction. The direction of a closed polyline is positive if it is drawn counter-clockwise and negative if it is drawn in a clockwise direction. Rectangles have a counter-clockwise direction if both the X- and Y-coordinate of the second point are higher or lower than the coordinates of the first point (= movement of the cursor is positive or negative along both X- and Y-axis). Rectangles have a clock-wise direction if the X- or Y-coordinate is higher or lower than the X- or Y-coordinate of the first point (= movement of the cursor is positive along the X-axis and negative along the Y-axis or vice versa).

The *Reverse Direction* option of the *Edit Polyline* tool changes the direction of a polyline.

Revision Clouds


Command: REVLOUD

A *Revision Cloud* is an open or closed polyline composed of arcs. Revisions clouds are used to for reviewing or redlining purposes to indicate the parts of the drawing that need to be adjusted or annotated.

Before you start drawing revision clouds, first define the *Arc length* and *Style*.

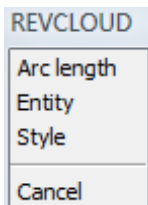
Drawing revision clouds

1. Do one of the following:

- Click the *Revision Cloud* tool button () on the *Draw* toolbar.
- Choose *Revision Cloud* in the *Draw* menu.
- Type *revision cloud* in the command bar, then press Enter.

The command bar reads: Specify start point or [Arc length/Object/Style] <Object>:

A prompt menu opens:



2. (option) To define the *Arc length*, do one of the following:

- Choose *Arc length* in the prompt menu.
- Type *A* in the command bar, then press Enter.

The command bar reads: Specify minimum length of arc <current minimum length>:

Do one of the following:

- Press Enter or right click to accept the current minimum length.
- Type a new value in the command bar, then press Enter.
- Define a new minimum length graphically by specifying two points.

The command bar reads: Specify maximum length of arc <current maximum length>:

Do one of the following:

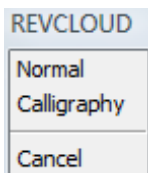
- Press Enter or right click to maximum length equal to the minimum length.
- Type a new value in the command bar, then press Enter.
- Define a new maximum length graphically by specifying two points.

3. (option) To define the revision cloud *Style*, do one of the following:

- Choose *Style* in the prompt menu.
- Type *S* in the command bar, then press Enter.

The command bar reads: Select arc style [Normal/Calligraphy] <current style>:

A prompt menu opens:



Do one of the following:

- Press Enter or right click to accept the current style
- Select a new style in the prompt menu.
- Type *N* or *C* in the command bar, then press Enter.

4. (option) To convert an existing polyline, arc or circle to a revision cloud, do one of the following:

- Press Enter, then select the entity you want to convert.
- Choose Object in the prompt box, then select the entity you want to convert.

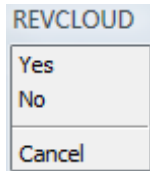
5. (option) To draw the revision cloud, click where you want to start and move the cursor. Each time the cursor movement exceeds the minimum length, an arc is added.

To stop do one of the following:

- Move the cursor over the start point to close the revision cloud.
- Right click.

The command bar reads: Reverse direction [Yes/No] <No>:

A prompt menu opens:



6. To create the revision cloud, do one of the following:
- To accept the revision cloud, right click.
 - To reverse the direction, type Y or choose *Yes* in the prompt menu.
 - To leave the command without creating the revision cloud, choose *Cancel* in the prompt menu or press the Esc key.



Revision Clouds: 1. Normal 2. Calligraphy 3. Reversed calligraphy (based on a rectangle)


Splines

Command: SPLINE

A spline is an open or closed smooth curve defined by a set of points. You can use splines to draw curved shapes, which cannot be drawn as a polyline.

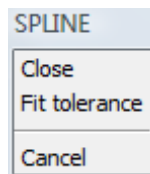
Drawing splines

1. Do one of the following

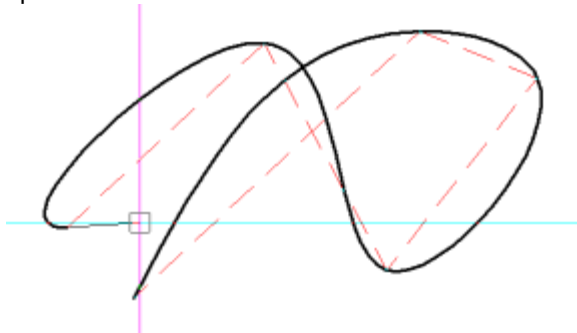
- Click the *Spline* tool button () on the *Draw* toolbar.
- Choose *Spline* in the *Draw* menu.
- Type *spline* in the command bar, then press Enter.

The command bar reads: First point of spline:

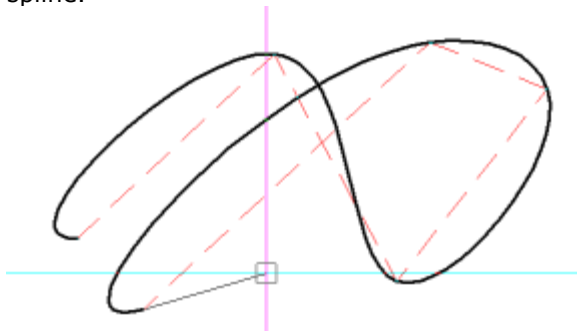
2. Specify the first point of the spline.
3. Define the second point of the spline.
The command bar reads: Close/Fit Tolerance/<next point> :
A prompt menu displays.



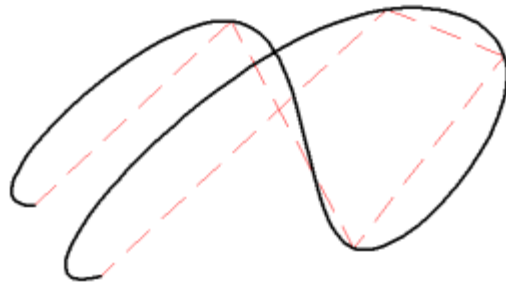
4. (option) Repeat step 3 to define more points.
5. Right click to stop adding more points.
The command bar reads: Select starting tangent point:
A line that is tangent to the spline displays dynamically from the start point of the spline.



6. Click to define the curve of the spline.
The command bar reads: Select ending tangent point:
A line that is tangent to the spline displays dynamically from the start end of the spline.



7. Click to conclude the spline command.



8. (option) Right click to restart.


- NOTES**
- The red dashed lines in the above images represent the polyline through the control points of the spline.
 - The *SPLFRAME* (Spline Frame) settings variable defines whether the control polygon displays for splines and spline-fit polylines.

Traces

Command: TRACE

Traces are 4-sided filled entities, created with two parallel sides. When you create a series of traces, they are L-connected to each other.

Drawing traces

1. Do one of the following
 - Click the *Trace* tool button () on the *Polygons* flyout of the *Draw* toolbar.
 - Choose *Trace* in the *Draw* menu.
 - Type *trace* in the command bar, then press Enter.

The command bar reads: Width of trace <current width>:
2. Do one of the following:
 - Right click to accept the current width.
 - Type a new current width in the command bar.
 - Define a new current width graphically by clicking two points.

The command bar reads: Start of trace:
3. Specify the start point of the trace.
The command bar reads: Next point:
4. Specify the end point of the first trace.
A line indicates the direction of the first trace. The direction of the second trace displays dynamically.
5. Do one of the following:
 - Right click to create the trace.
 - Specify the endpoint of the next trace.
6. (option) Repeat step 5 to keep adding traces.

NOTE When Fill Mode is turned off, all filled entities, such as traces, display and print as outlines.


Wipeouts

Command: WIPEOUT

A *wipeout* is a polygonal area that hides the underlying objects using the current background color. Most commonly wipeouts are used to mask part of a drawing for clarity. For example, you may want to add text to a complicated part of a drawing. A wipeout could be used then to mask an area behind the text to increase the readability of the text.

The *Frames* option of the *wipeout* command determines whether the edges of all wipeout entities in the current drawing are displayed or hidden.


To create a wipeout using points

1. Do one of the following:
 - Click the *Wipeout* tool button  on the *Draw* toolbar.
 - Choose *Wipeout* in the *Draw* menu.
 - Type *wipeout* in the command bar, then press Enter.

The command bar reads: Frames/Polylines/<Start point>:
2. Specify the first vertex of the wipeout.
The command bar reads: Next point.
3. Specify the second vertex of the wipeout.
The command bar reads: Undo/<Next Point>:
4. Do one of the following:
 - Specify the third vertex of the wipeout.
 - Type *U*, then press Enter or choose *Undo* in the context menu to delete the previous vertex.

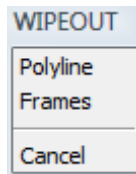
The command bar reads: Undo/Close/<Next Point>:
5. Do one of the following:
 - Specify the next vertex of the wipeout.
 - Type *U*, then press Enter or choose *Undo* in the context menu to delete the previous vertex.
 - Type *C*, then press Enter or choose *Close* in the context menu to create the wipeout.
 - Right click to create the wipeout.
6. Repeat step 5 to create more vertices of the wipeout.

To create a wipeout from a polyline

1. Do one of the following:
 - Click the *Wipeout* tool button  on the *Draw* toolbar.
 - Choose *Wipeout* in the *Draw* menu.
 - Type *wipeout* in the command bar, then press Enter.

The command bar reads: Frames/Polylines/<Start point>:


The *Wipeout* context menu displays:



2. Do one of the following:
 - Choose *Polyline* in the context menu.
 - Type *P*, then press Enter.

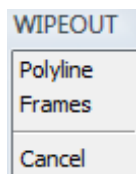
The command Bar reads: Select a closed polyline:
3. Click a closed polyline.
 Only polylines composed of line segments can be selected.
 The command bar reads: Delete selected polyline? Yes/<No>:
4. Do one of the following:
 - Type *Y*, then press Enter or choose *Yes* in the context menu to create the wipeout and delete the selected polyline.
 - Press Enter or choose *No* in the context menu, to create the wipeout and delete the selected polyline.

To control the display of the edges of wipeouts

1. Do one of the following:
 - Click the *Wipeout* tool button () on the *Draw* toolbar.
 - Choose *Wipeout* in the *Draw* menu.
 - Type *wipeout* in the command bar, then press Enter.

The command bar reads: Frames/Polylines/<Start point>:

The *Wipeout* context menu displays:



2. Do one of the following:
 - Choose *Frames* in the context menu.
 - Type *F*, then press Enter.

The command bar reads: Enter display mode: OFF/<ON>:
3. Do one of the following:
 - Choose *OFF* in the context menu or type *off*, then press Enter, to hide the edges of all wipeouts in the drawing.
 - Choose *ON* in the context menu or type *on*, then press Enter, to display the edges of all wipeouts in the drawing.

Mechanical

Components and Component Inserts

Assembly modeling consists in using mechanical components to group geometric entities in hierarchies, representing the structure of a designed product. Assembly modeling is commonly used in Mechanical Computer-Aided Design (MCAD) applications, because it simplifies design of complex products containing thousands and millions of geometric entities.

A mechanical component is a named group of entities. A particular example of a mechanical component is any .dwg file. Another example of a component is a standard hardware. Users can define their own components. A component can be inserted one or several times into one or more other components.

A component insert is a named entity, which default name is composed by the name of the component and the serial number of the insert. An assembly is a mechanical component with other components inserted in it.

An assembly with inserted components (which in turn can be assemblies with other inserted components) forms a tree that is called a mechanical structure of the drawing. This tree is displayed in the **Mechanical Browser**.

Mechanical components are stored in .dwg files as custom objects. The geometric entities represented particular components can be rendered and edited in other dwg-based applications.

Users can create mechanical components from scratch or add them to existing .dwg

Commands and Toolbars

Tools to work with mechanical components are available on the *Mechanical* toolbar and its *Mechanical Visualization* flyout toolbar:



Icon	Command	Description
	BmBrowser	Toggles the visibility of the <i>Mechanical Browser</i> window.
	BmUpdate	Reloads all referenced components from external files and updates BOM tables.
	BmNew	creates a new mechanical component as a new drawing file.
	BmInsert	Inserts an existing mechanical component into the current drawing
	BmForm	Creates a new mechanical component and inserts it into the current drawing.
	BmDissolve	Dissolves a mechanical component inserted into the current drawing.
	BmMech	Convert the current drawing into a mechanical main component.
	BmHardware	Inserts a standard hardware solid as a mechanical component.
	BmHide	Hides a mechanical component or subcomponent in the current drawing.
	BmShow	Shows a previously hidden mechanical component or subcomponent in the current drawing.
	BmVStyle	Applies a visual style to a mechanical component
	BmBom	Inserts the Bill-of-Materials (BOM) table in the current drawing

	BmXConvert	Converts X-Hardware solids in the current drawing to mechanical components.
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Mechanical Browser

Commands: BMBROWSER, BMHARDWARE

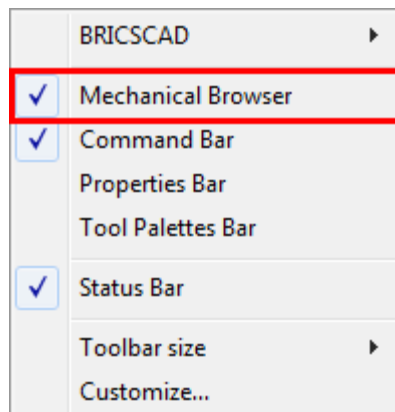
The Mechanical Browser allows navigation on the hierarchy of mechanical components for the current drawing and calling different commands for the inserts. 2D and 3D Constraints also display in the Mechanical Browser.

With the mechanical browser you can:

- click a component to highlight its geometric entities in the drawing window
- click a component to change its name
- right click a component to run a command for it
- edit 2D and 3D constraints
- insert a standard hardware parts as a mechanical component in the current drawing.

To open the Mechanical Browser do one of the following:

- Choose *Mechanical Browser* in the *Mechanical* menu.
- Place the cursor on a toolbar, then right click and choose *Mechanical Browser* in the context menu.



Bill Of Materials

Command: BMBOM

Engineering Bill of Materials (BOM) is a list of unique mechanical components used in a drawing. The list is represented as a BricsCAD table entity (see Tables), and can be placed at any location in the drawing area. Each entity has its serial number (starting with 1), name, and the number of occurrences.

Bill of materials NV-ENT-Railway_inspection_car		
No	Part	Quantity
1	BBRG-00x40 MM	4
2	ISO 1035-FLx1.5 X 15x10	4
3	ISO 2009xM8 X 1x10	8
4	ISO 4014xM10 X 1.25x100	2
5	ISO 4014xM16 X 2x100	8
6	ISO 4034xM10 X 1.25	2
7	ISO 4034xM16 X 2	8
8	NV-AX-005-01-axis_2	1
9	NV-AX-010-01-axis_1	1
10	NV-BRK-005-01-bracket	4
11	NV-RD-005-01-rod_1	1
12	NV-RD-005-01-rod_1	1
13	NV-SBS-005-01-platform	1
14	NV-SI-005-01-bench	2
15	NV-SU-005-01-bearing	1
16	NV-WH-005-01-wheel	4

There are two options for BOM: listing top-level components only or bottom-level (parts) only.

Organizing Geometry in Components

There are two main ways to design mechanical products: top-down and bottom-up approaches. It is also possible to add mechanical structure to existing geometry (designed in BricsCAD or imported from another source).

BmNew command is called to create a new document for a new mechanical component.

Bottom-Up Design

With bottom-up approach users start to design a product with detailing its low-level component. These components are then grouped into higher-level components. The process is continued till the very top component (corresponding to the product itself) will be designed.

The main command for bottom-up design is **BmInsert**, which is used to insert an existing component into the current drawing.

Top-Down Design

With top-down approach users start to design a product with defining (but not detailing) its first-level components. Each component is then refined in more details by defining its sub-components. The process is continued till all low-level components will be defined and detailed.

BmForm command allows to create a new subcomponent for the existing component.

Adding Mechanical Structure to Existing Geometry

BricsCAD users can add component hierarchy to any existing drawing – created in BricsCAD or imported from a different CAD format.

BmForm command is called to convert a selected set of geometric entities to a subcomponent of the current drawing.

Changing Mechanical Structure

During design process users may realize than a particular component should be moved up or down in the component hierarchy.

BmForm command forms a new subcomponent for the current drawing that will contain a selected set of component.

BmDissolve command can be called to delete a particular component and move all its subcomponents into the level, where the whole component was inserted.

Using Standard Hardware

Standard hardware (screws, nuts, washers, bearings, etc.) can be considered as a mechanical component.

BricsCAD supplies a huge library of standard hardware (more than 30,000 details of different size). Users can significantly increase the design performance by using standard parts.

BmHardware command is called to insert a standard part into the existing component.

Assembly Constraints

Assembly constraints are ordinary **3D Constraints**, which link faces and edges of bodies that represent the components of an assembly. 3D Constraints can link not only top-level components, but also any their subcomponents.

Component Visibility

A particular component of a complex product can be invisible due to other components surrounding it. For example, a piston is invisible inside a cylinder block. To make such component visible users may hide some other components.

Users can hide any subcomponent inserted into the current drawing or any its subcomponent on any level of nesting.

There is also a possibility to make a component transparent – to visualize it in wireframe model.

BmHide command is called to hide a particular subcomponent.

BmShow command is called to show a hidden subcomponent.

Assembly Design and Kinematic Analysis Tutorial



Watch the movie about Kinematic Analysis on the Bricsys channel on YouTube.

The source files for this tutorial are installed in the *[BricsCAD Installation folder]\Samples\Mechanical\piston* folder.

Assembly design is a typical application where constraints are commonly used. Since V13, BricsCAD Platinum allows users to assemble mechanisms and run kinematic analysis.

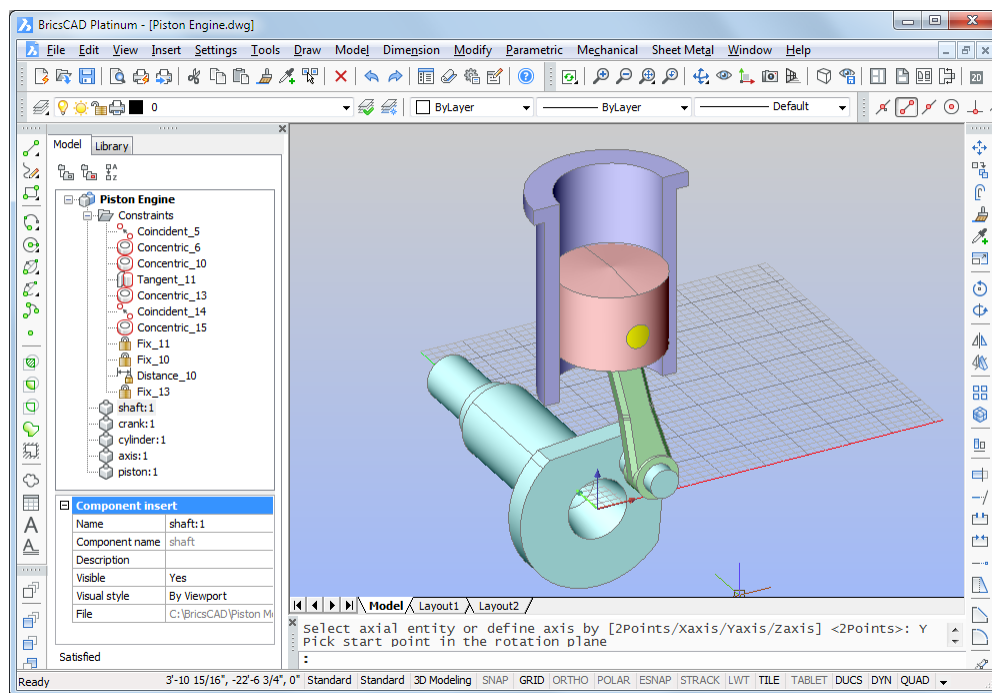
This tutorial shows you how to start working with assemblies in BricsCAD. You will learn how to assemble the different parts in a 3D model, which lets you control the final result.

There are two approaches in MCAD (**M**echanical **C**omputer-**A**ided **D**esign) to build an assembly: top-down and bottom-up.

In the top-down approach, you start with an empty assembly and the geometry of each component is then created, one by one, in the assembly.

In the bottom-up approach, each component is first created as a single entity. All components are then inserted into the assembly. The position of each component is controlled through 3D constraints.

We will now assemble a simplified - in comparison with a real-world engine - model of a piston engine.



Step 1: Preparing for the exercise

1. You need a *BricsCAD Platinum* license for this tutorial.
2. Make sure the *Mechanical* and *3D Constraints* toolbars are open.

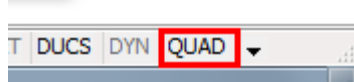




To open a toolbar: move the cursor over an open toolbar, then right click and select the required toolbars from the toolbar list.

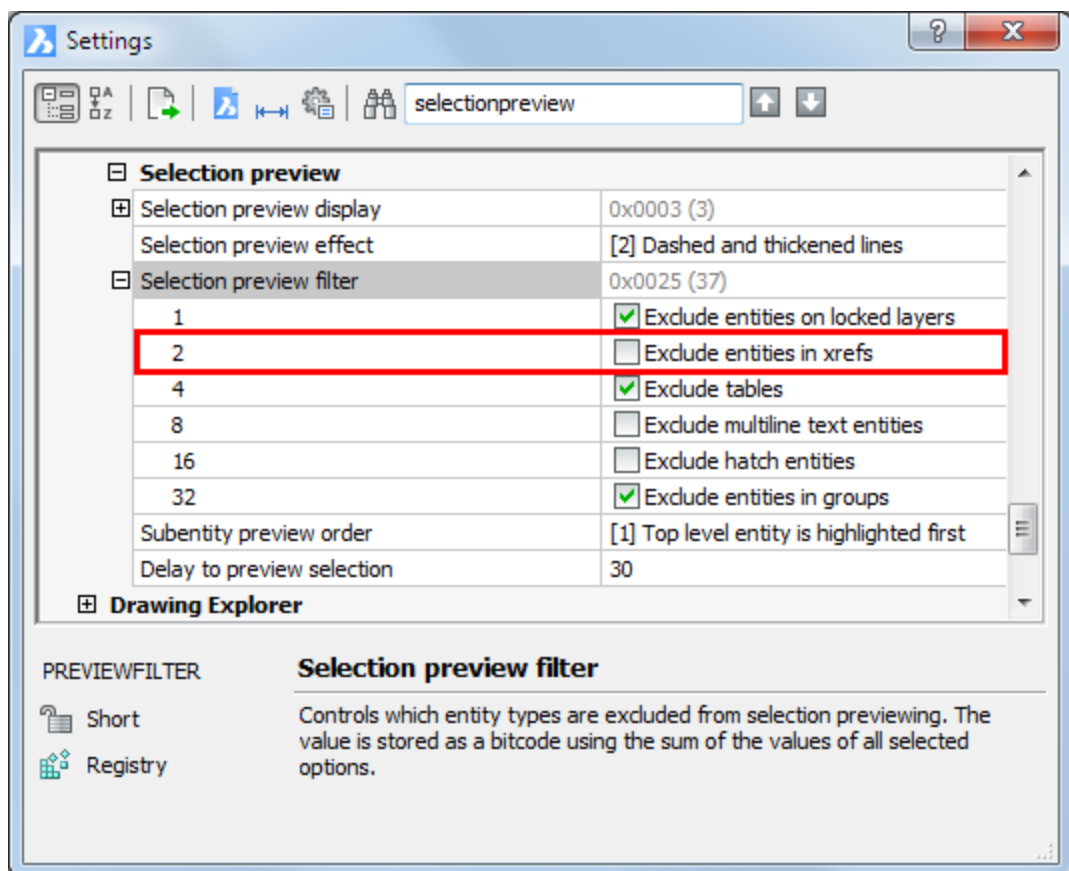
Toolbars that are already open are marked in the list.


3. Make sure that the *Quad* cursor menu is active.

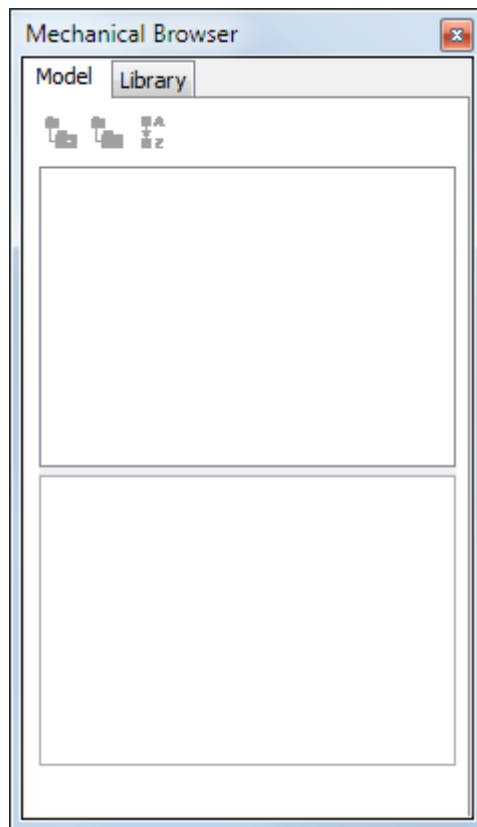


Click the *Quad* field in the Status bar if necessary.


4. Make sure that the *Exclude entities in Xrefs* option of the PREVIEWFILTER system variable is switched off.

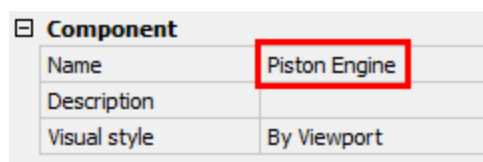


5. Open the *Mechanical Browser* dialog box.
Do one of the following:
 - Click the *Mechanical Browser* tool button () on the *Mechanical* toolbar.
 - Choose *Mechanical Browser* in the *Mechanical* menu.
 - Type *bmbrowser* in the command bar.



Step 2: Creating the assembly drawing


1. Click the *New Component* tool button () on the *Mechanical* toolbar.
A new drawing of the name *Component_1.dwg* is created and *Component_1* is added in the *Mechanical Browser*.
2. In the bottom part of the *Mechanical Browser*, click the *Name* field and rename the main component: *Piston Engine*.

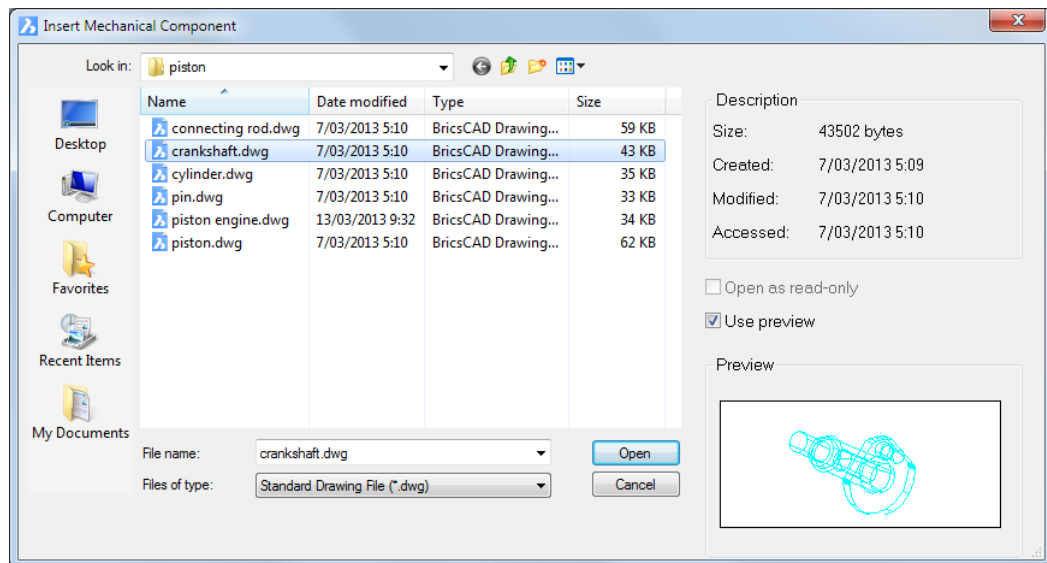


The drawing name automatically matches the main component name.

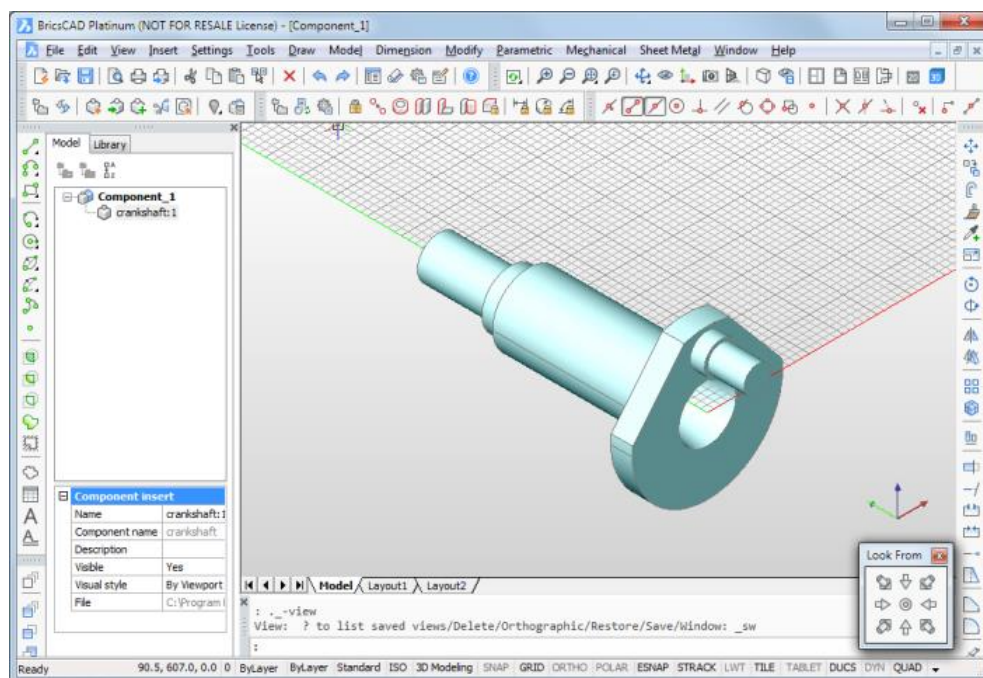
3. Save the drawing.

Step 3: Adding the components

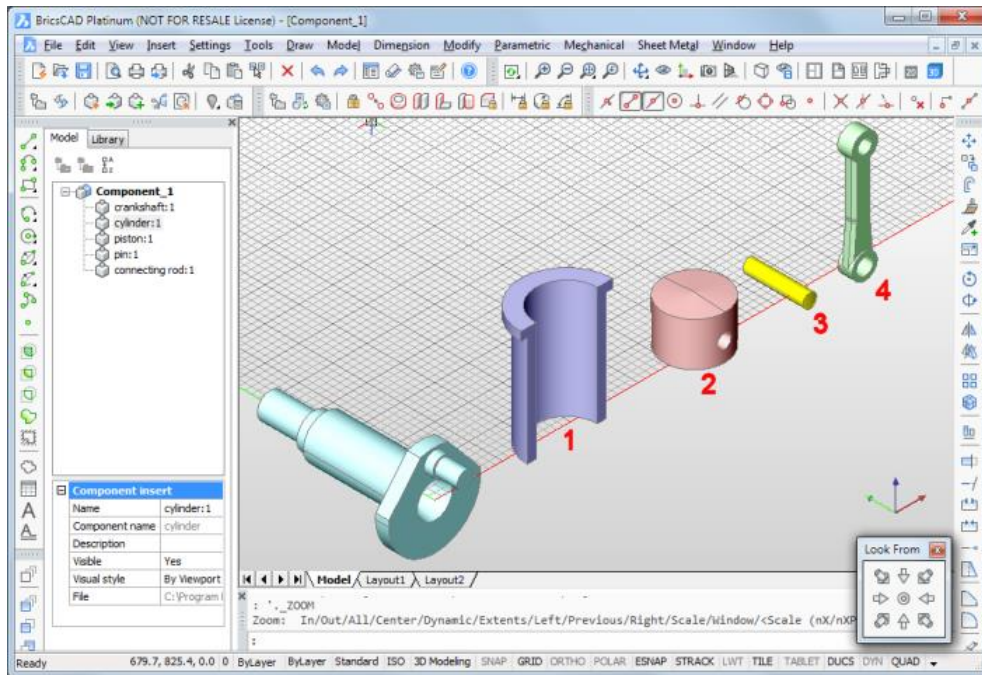
1. Click the *Insert Component* tool button () on the *Mechanical* toolbar.
The *Insert Mechanical Component* dialog displays:
Open the *[BricsCAD Installation folder]\Samples\Mechanical\piston* folder.



2. Select the *crankshaft.dwg*, then double click or click the *Open* button.
The component is attached to the cursor with its origin point.
Dynamic dimensions display: Distance from the origin of the current coordinate system and angle from the X-axis.
3. We will consider the crankshaft as the anchor component of the assembly, and therefore insert it at the origin (0,0,0).
Please make sure the insertion point is exactly at 0,0,0, otherwise the kinematic analysis in step 6 might fail.



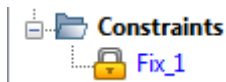
4. Use the tools on the *Look From* toolbar to adjust the view orientation.
5. Repeat the previous steps to insert the other components: *cylinder* (1), *piston* (2), *pin* (3) and *connecting rod* (4).



Step 4: Positioning the components

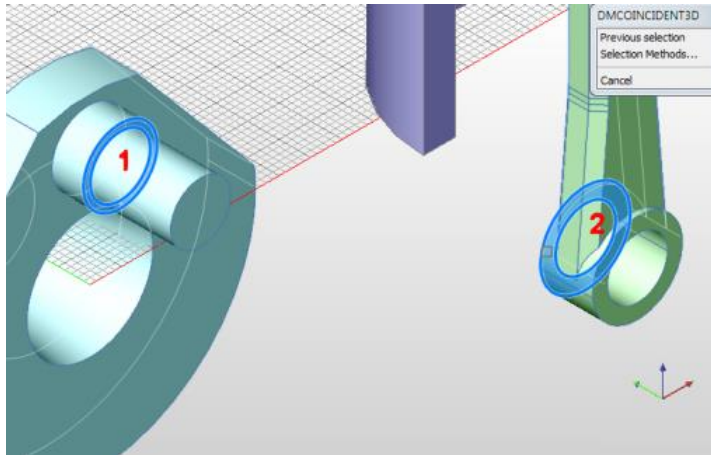
1. Click the *Fix* tool button (🔒) on the *3D Constraints* toolbar.
The command bar reads: Select an edge, face or 3D solid:
2. Move the cursor over the crankshaft, then do one of the following:
 - Hit the TAB key until the whole solid is selected.
 - Press and hold the Shift key, then hit the TAB key.


The position of the crankshaft is now locked, as indicated in the *Constraints* tree in the *Mechanical Browser*.

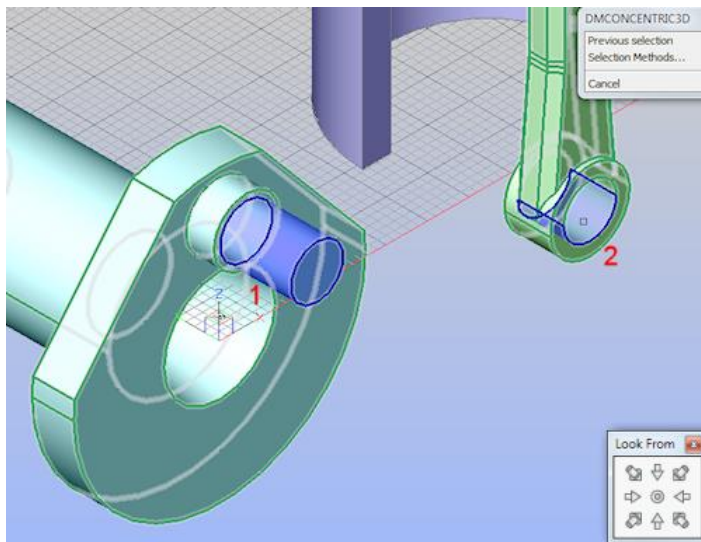


Clicking the constraint highlights the solid, face or edge it applies to.

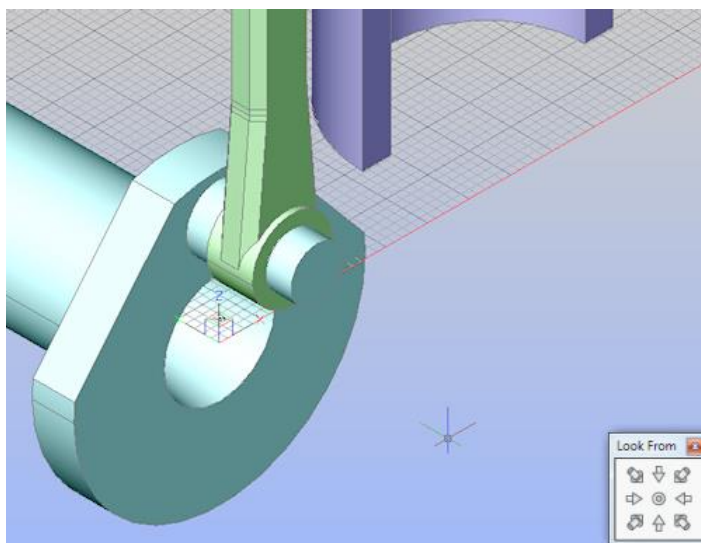
3. Click the *Coincident* tool button (○) on the *3D Constraints* toolbar.
The command bar reads: Select a pair of subentities:
4. Move the cursor over two planar faces of the crankshaft (1) and the connecting rod (2) as indicated in the image below.
Hit the TAB key until the correct face highlights, then click and move the cursor to the second face.




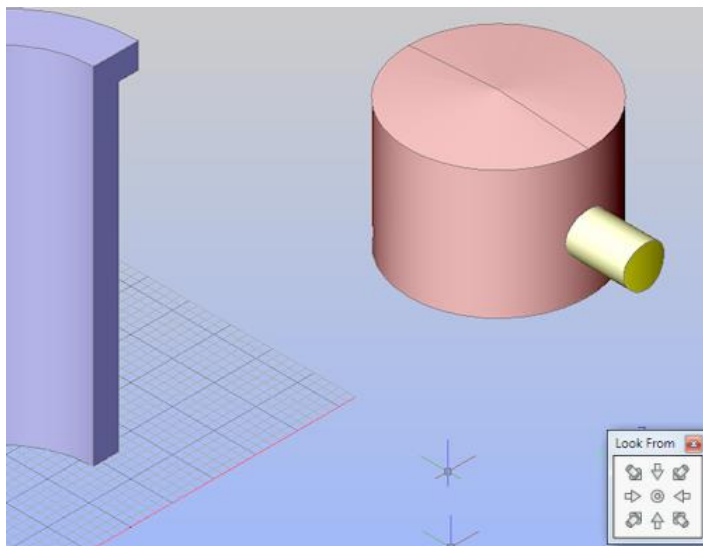
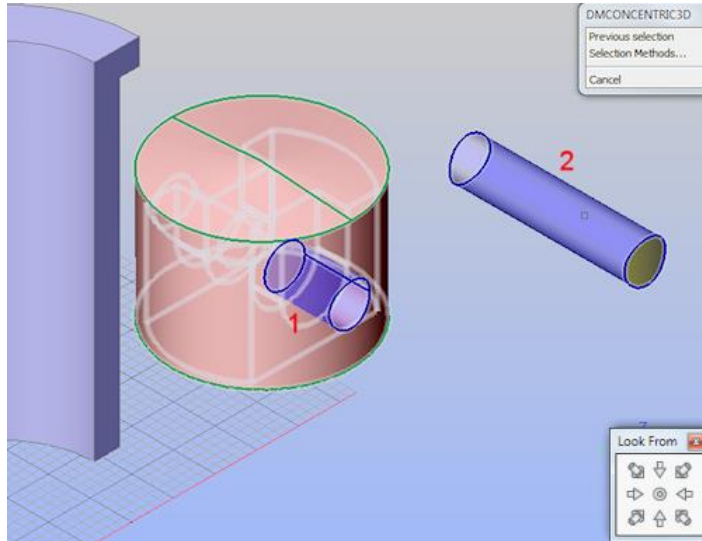
5. Click the *Concentric* tool button () on the *3D Constraints* toolbar.
The command bar reads: Select a pair of subentities:
6. Move the cursor over two cylindrical faces of the crankshaft (1) and the connecting rod (2) as indicated in the image below.
Hit the TAB key until the correct face highlights, then click and move the cursor to the second face.




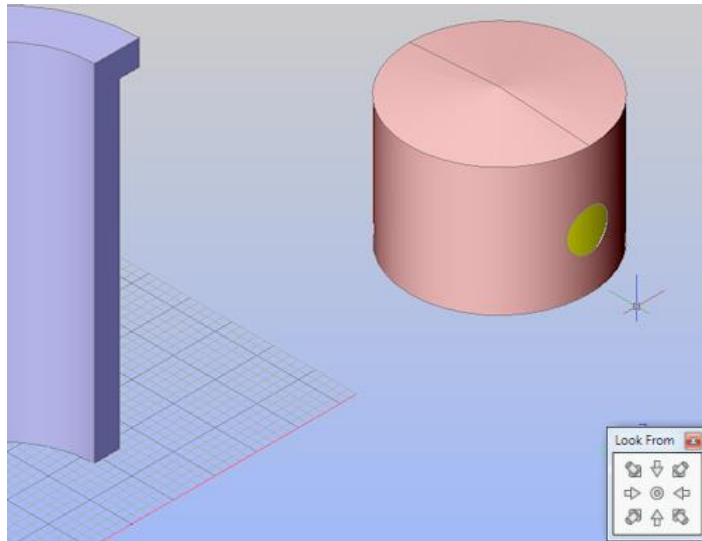
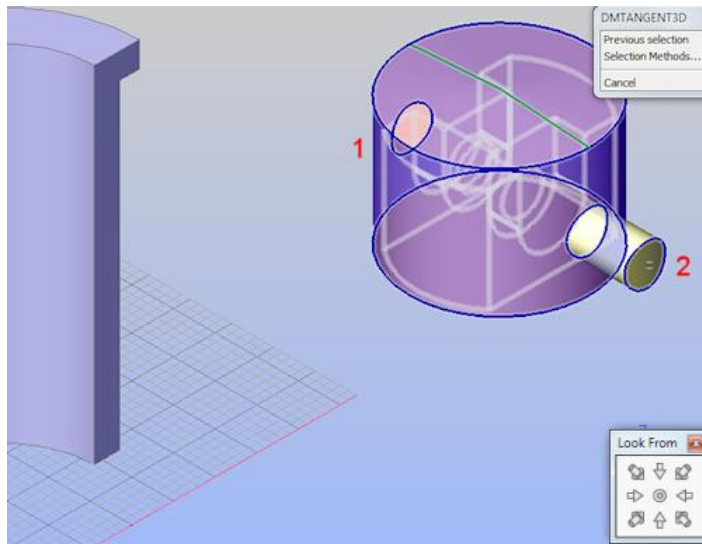
The connecting rod is automatically moved to the right position according to our design intent.




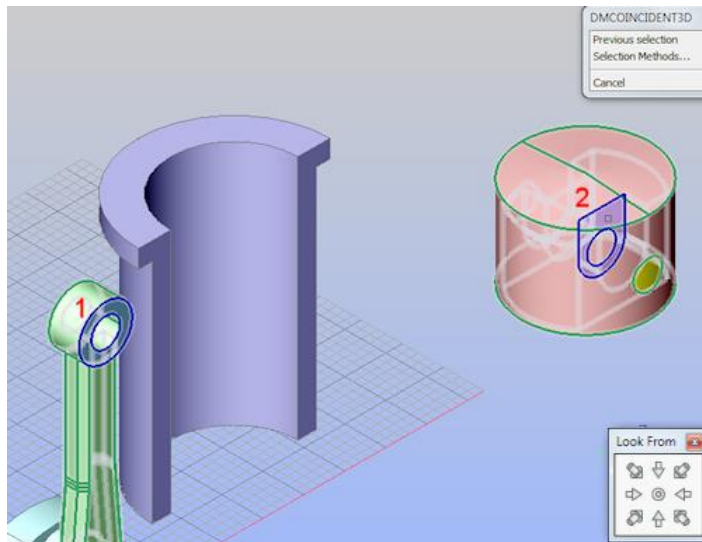
7. Click the *Concentric* tool button () on the *3D Constraints* toolbar.
The command bar reads: Select a pair of subentities:
8. Move the cursor over two cylindrical faces of the piston (1) and the pin (2) as indicated in the image below.
Hit the TAB key until the correct face highlights, then click and move the cursor to the second face.




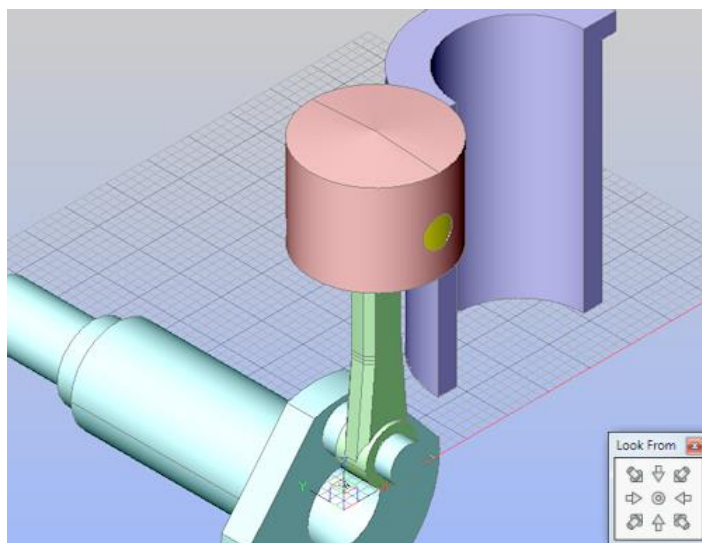
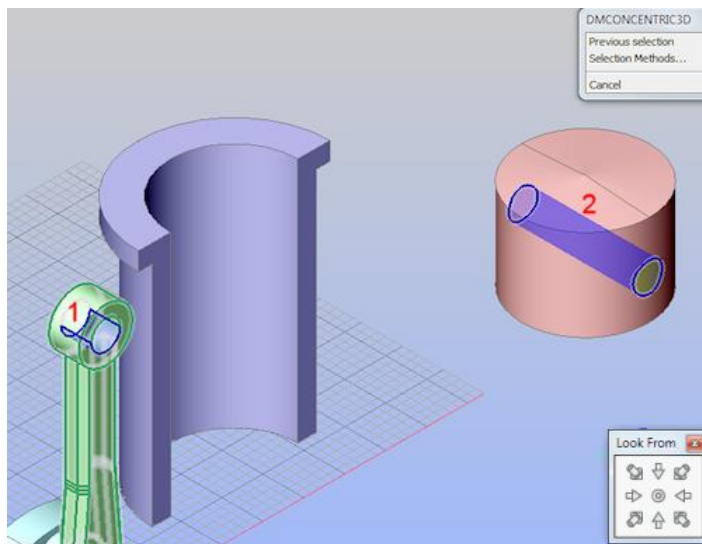
9. Click the *Tangent* tool button () on the *3D Constraints* toolbar.
The command bar reads: Select a pair of subentities:
10. Move the cursor over two cylindrical faces of the piston (1) and the pin (2) as indicated in the image below.
Hit the TAB key until the correct face highlights, then click and move the cursor to the second face.




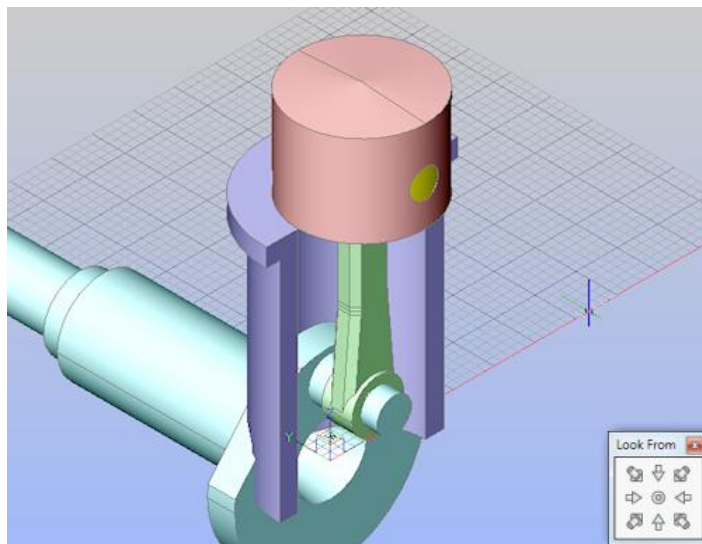
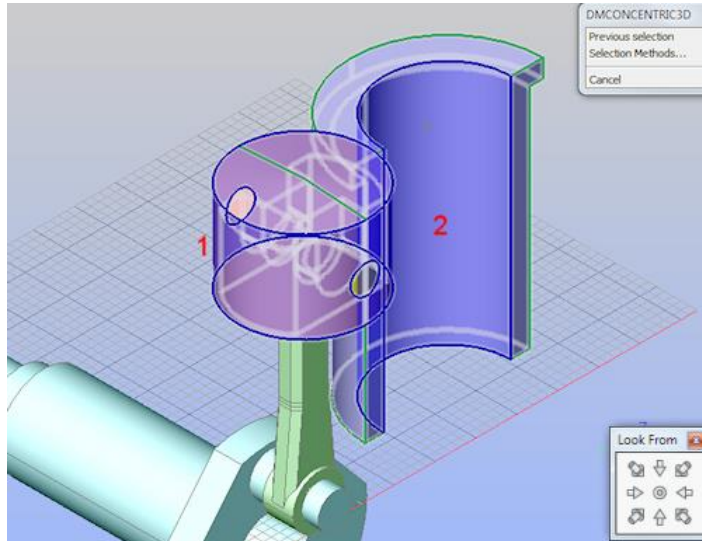
11. Click the *Coincident* tool button () on the *3D Constraints* toolbar.
The command bar reads: Select a pair of subentities:
12. Move the cursor over two planar faces of the connecting rod (1) and the piston (2) as indicated in the image below.
Hit the TAB key until the correct face highlights, then click and move the cursor to the second face.




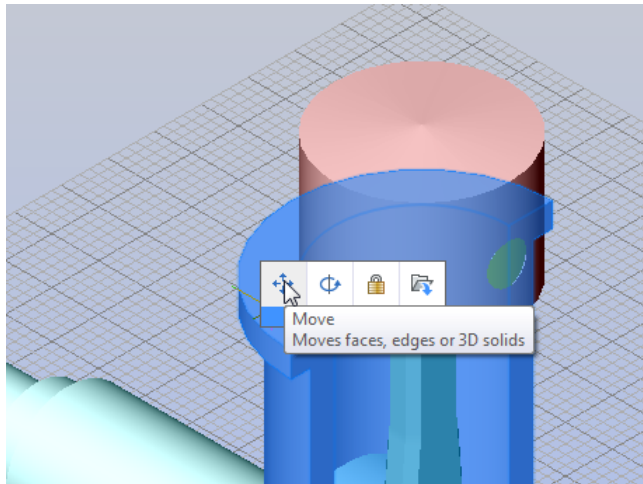
13. Click the *Concentric* tool button () on the *3D Constraints* toolbar.
The command bar reads: Select a pair of subentities:
14. Move the cursor over two cylindrical faces of the connecting rod (1) and the pin (2) as indicated in the image below.
Hit the TAB key until the correct face highlights, then click and move the cursor to the second face.



15. Click the *Concentric* tool button () on the *3D Constraints* toolbar.
The command bar reads: Select a pair of subentities:
16. Move the cursor over two cylindrical faces of the piston (1) and the cylinder (2) as indicated in the image below.
Hit the TAB key until the correct face highlights, then click and move the cursor to the second face.

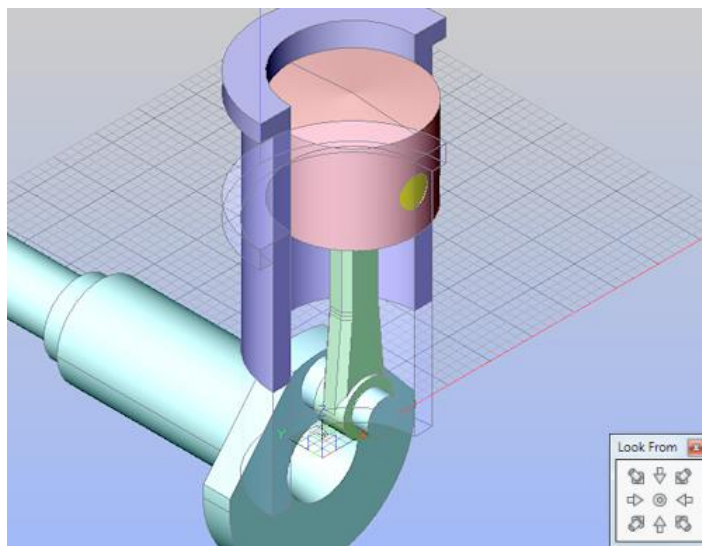


17. After applying the previous constraint it might be necessary to correct the position of the cylinder.
Move the cursor over the cylinder, then choose *Move* () in the *Quad* cursor menu.



The command bar reads: Enter base point <0,0,0>:


18. Use an entity snap (e.g. *Endpoint*) to specify the base point for the move operation on the cylinder.
The cylinder now moves dynamically with the cursor.
19. Hold down the Shift key to move the mouse cursor along the Z-axis.
Click to position the cylinder as shown in the image below.

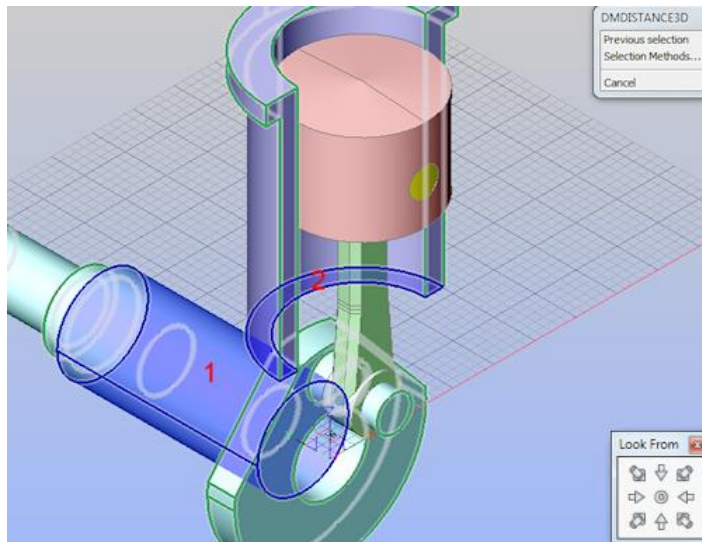


Step 5: Preparing for the kinematic analysis

The piston is correctly positioned inside the cylinder and connected to the crankshaft by the connecting rod.

We will now fix the distance between the cylindrical face of the crankshaft and the lower face of the cylinder by applying a dimensional constraint.

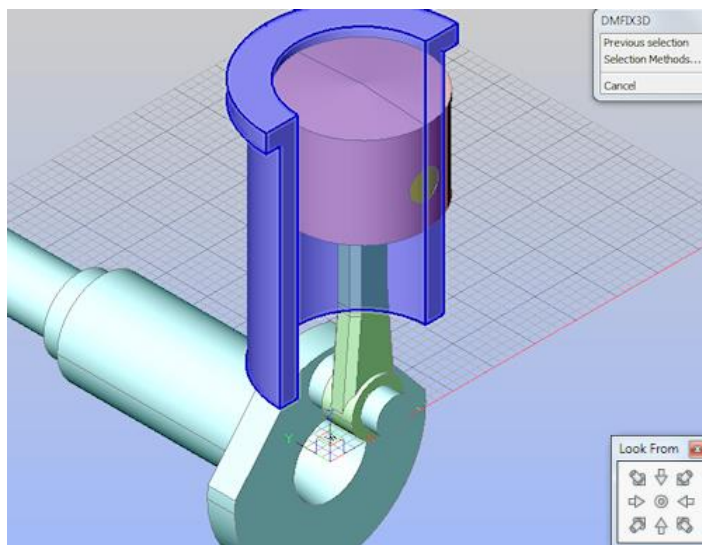
1. Click the *Distance* tool button () on the *3D Constraints* toolbar.
The command bar reads: Select a pair of subentities:
2. Select the cylindrical face of the crankshaft (1) and the lower planar face of the cylinder (2) as indicated in the image below:
The command bar reads: Enter distance value <133.622>:
A dynamic distance entry field displays.



3. Type 110 in the dynamic entry field.
The lower face of the cylinder is now fixed at a distance of 110 from the cylindrical face of the crankshaft.

We will fix the cylinder at its current position:

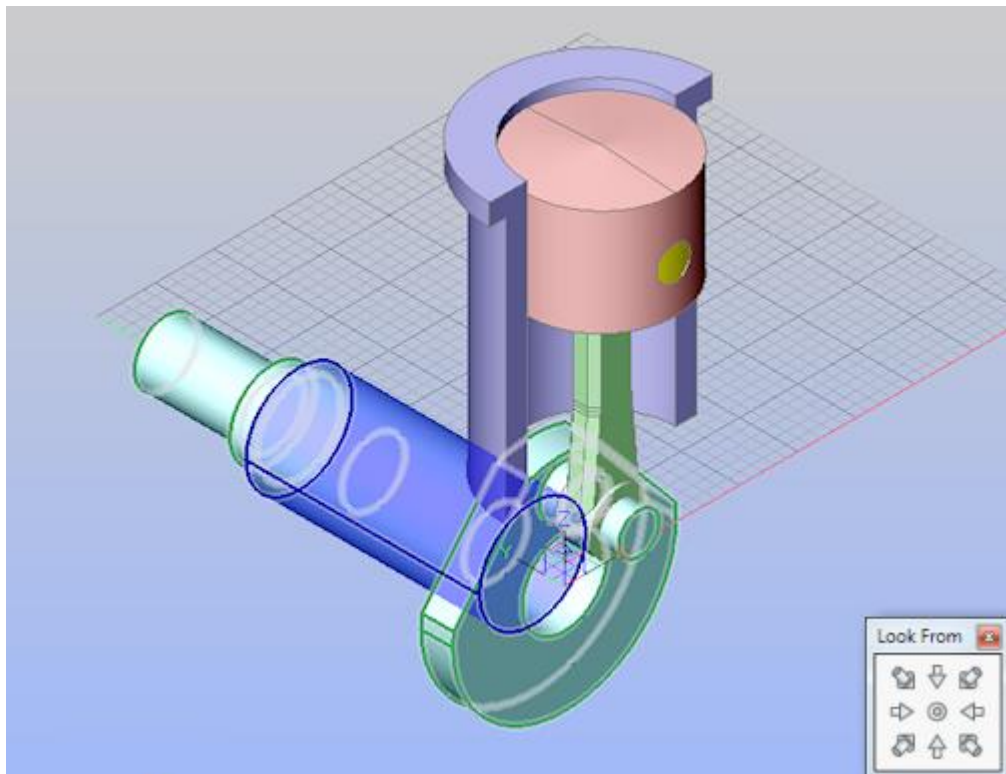
4. Click the *Fix* tool button (🔒) on the *3D Constraints* toolbar.
The command bar reads: Select an edge, face or 3D solid:
5. Move the cursor over the cylinder, then hit the TAB key until the body of the cylinder highlights as a whole (see image below).



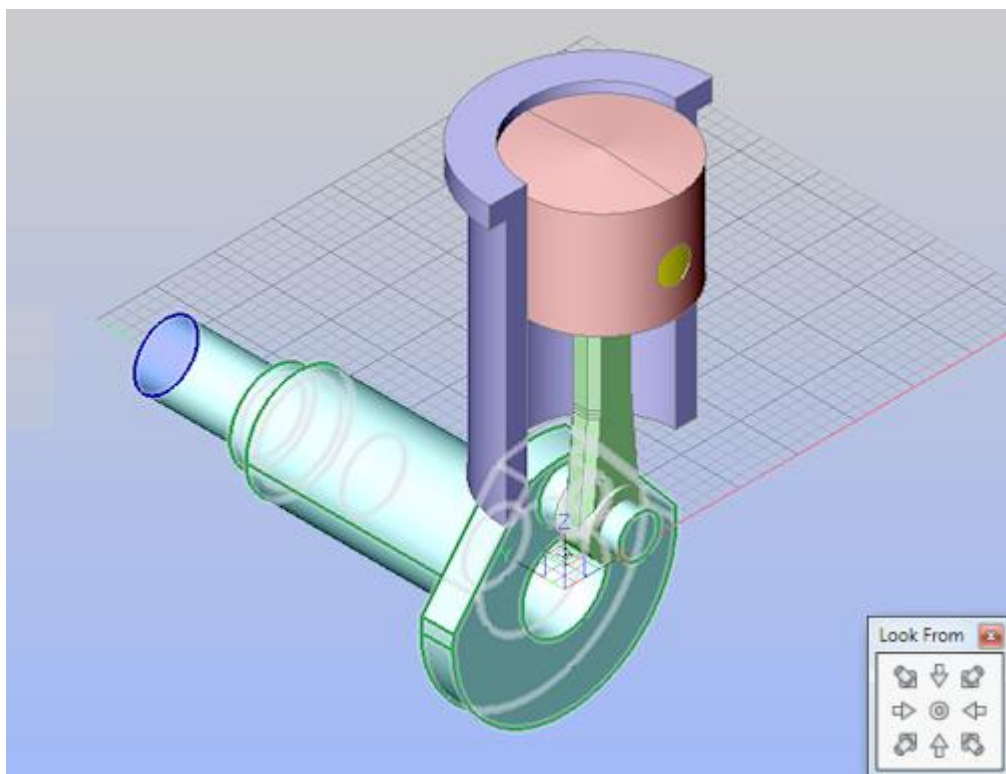
In step 4.1 we have applied a fixation constraint to the body of the crankshaft, which removes all degrees of freedom from the crankshaft. To make it possible to rotate the crankshaft around its axis, it is necessary to remove this constraint and replace it by two fixation constraints: one to the cylindrical face of the crankshaft and one to its planar outer faces.

6. Click the *Fix* tool button (🔒) on the *3D Constraints* toolbar.
The command bar reads: Select an edge, face or 3D solid:

7. Move cursor over the cylindrical face of the crankshaft, then click (see image below)



8. Repeat the previous steps for the planar outer face of the crankshaft (see image below).

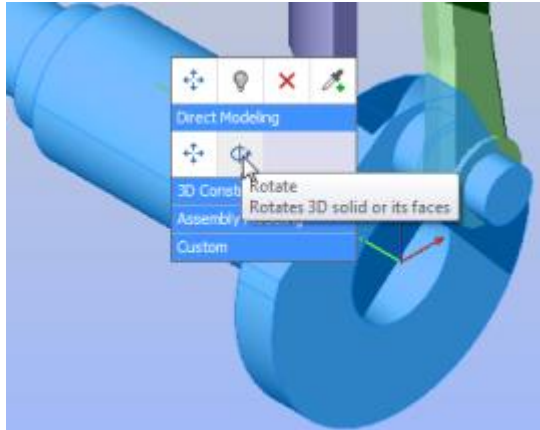


9. Select the *Fix* constraint applied in step 4.1 in the *Constraints* tree in the *Mechanical Browser*. Right click and select *Delete* in the context menu.

Step 6: Kinematic Analysis

In this assembly the crankshaft is the driving component. To run the kinematic analysis we will rotate the crankshaft around its axis, which in this case is the Y-axis of the WCS.

1. Move the cursor over the crankshaft, then choose *Rotate* (⌂) in the *Quad* cursor menu

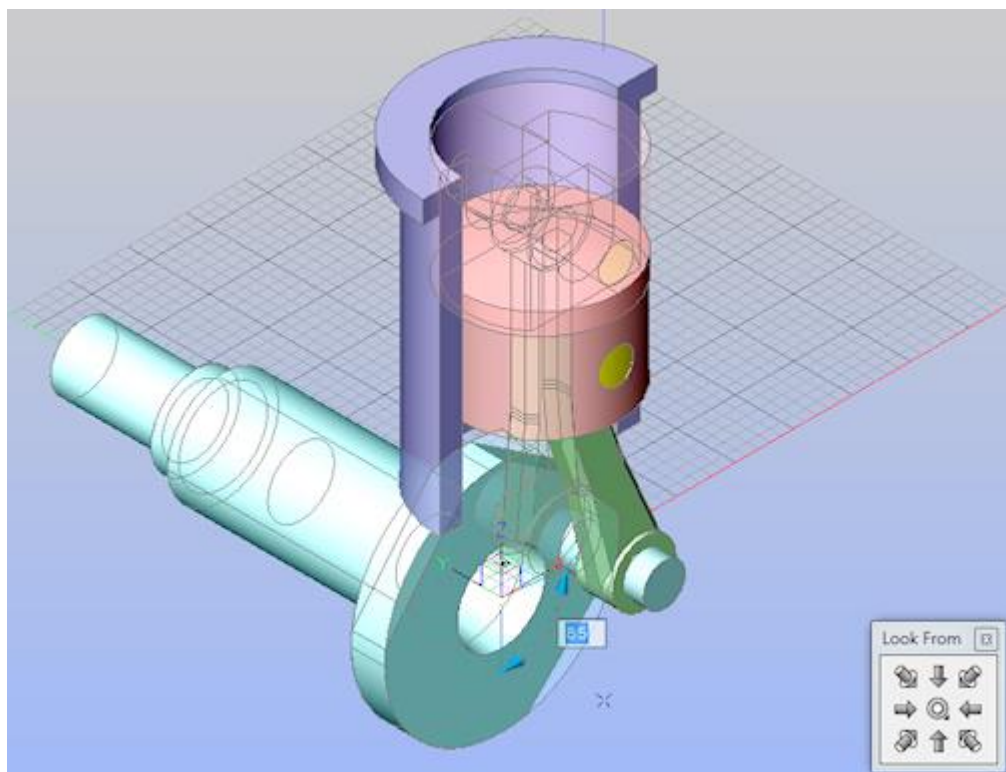


The command bar reads: Select axial entity or define axis by [2Points/Xaxis/Yaxis/Zaxis] <2Points>:

2. Do one of the following:
 - Type *Y*, then press Enter.
 - Choose *Y axis* in the prompt menu.

The command bar reads: Pick start point in the rotation plane:

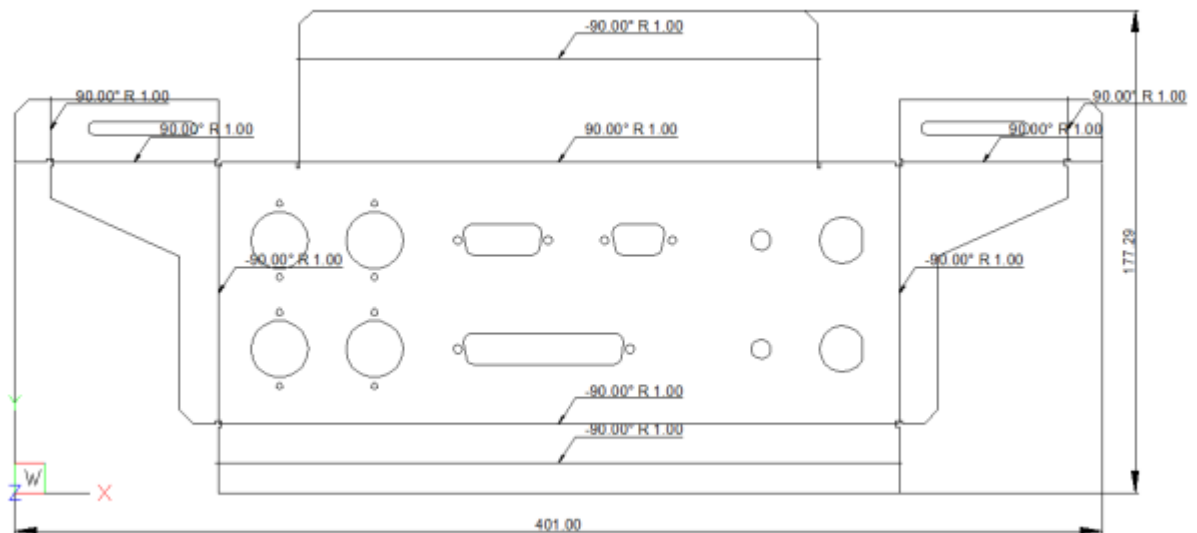
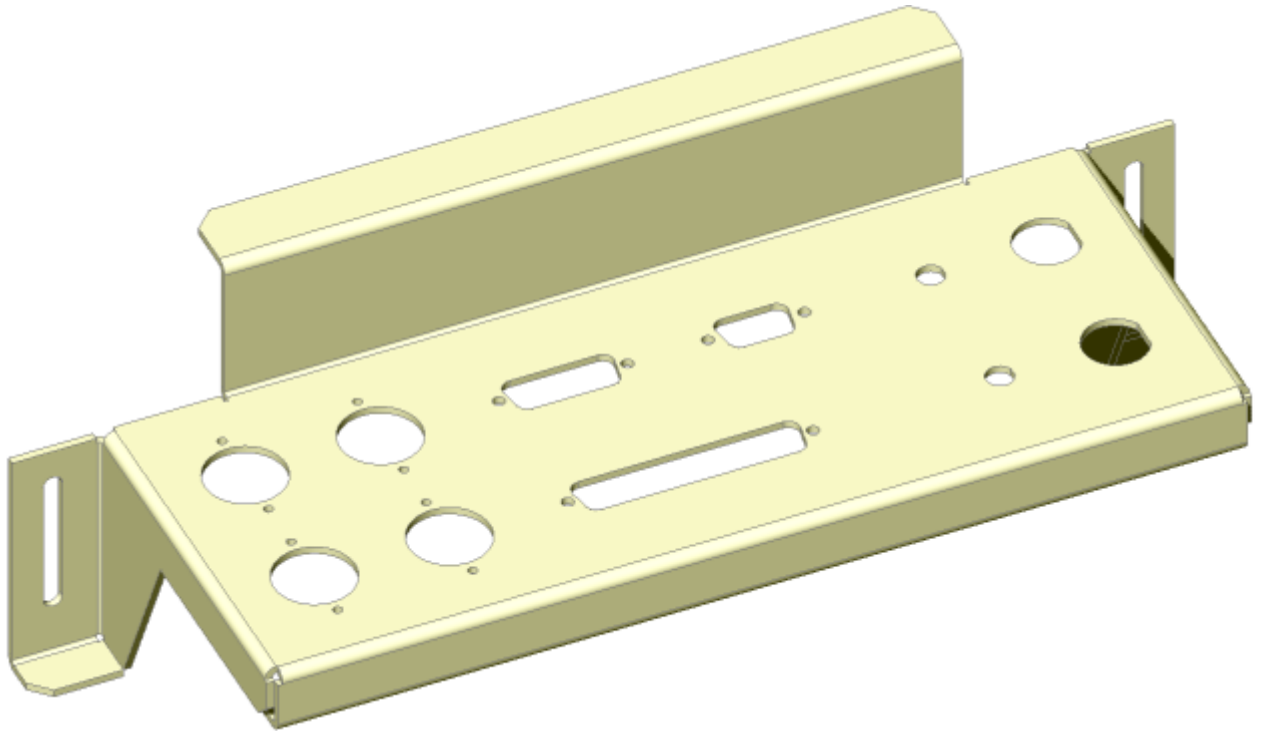
3. Pick a point in the drawing, then move the mouse to define the rotation angle. The piston and the connecting rod move dynamically along with the rotation of the crankshaft, while the cylinder stays at its fixed position.



SheetMetal

Commands: SmFlangeBase, SmFlangeEdge, SmFlangeRotate, SmFlangeConnect and SmUnfold

Sheet metal design allows you to model sheet metal parts and generate their unfolded representations with manufacturing information.



You can create complex sheet metal parts with BricsCAD easily and rapidly, because the design process is different from manufacturing process. Do not think in terms of a planar sheet that should be cut and bent, but model your part directly as you create solid bodies with Direct Modeling tools.

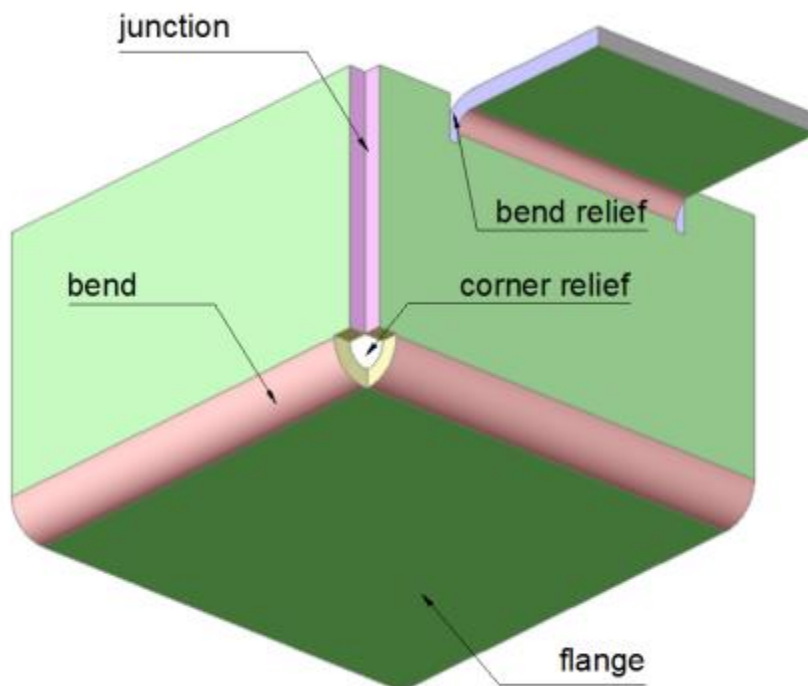
Creating a sheet metal part in BricsCAD consists of the following basic steps:

- Create a base flange from a closed planar profile.
- Create flanges:

- Pull its edges to create additional edge flanges with bends.
- Create partial flanges with automatic creation of the corresponding bend reliefs.
- Pull several adjacent edges to create several flanges at once with automatic creation of corner reliefs and junctions between them.
- Make holes by drawing closed profiles on faces of the flanges and pushing them through the flanges.
- Adjust material thickness and bend radius using pre-defined parameters.
- Apply Direct Modeling operations and 3D constraints to further adjust your design while always maintaining the design intent in terms of sheet metal features.
- Automatically generate unfolded representation of your part and send it to a CAM system by exporting the corresponding 2D drawing with bend annotations in a .dwg or .dxf file.

Sheet Metal Features

Form features are smart regions (groups of faces) of your 3D part. Each feature maintains specific spatial and parametric relationship between its faces and some adjacent faces. Form features allow you to embed design intent into your model. You do not create features by hand, they are created automatically depending on a particular geometric operation you apply.



Flange

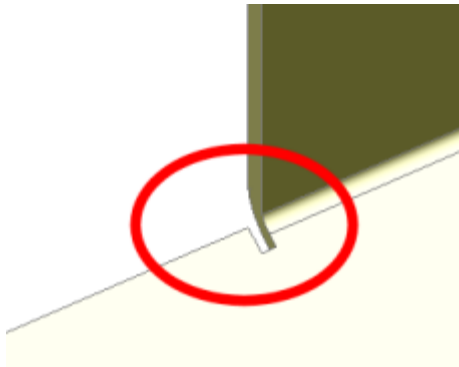
Main feature of any sheet metal part is a flange, which consists of two parallel planar faces. This distance between them is equal to the material thickness. When you modify the model, this distance relation is always maintained automatically. Other faces, which are adjacent to flange faces and do not belong to bends are called thickness faces. They are always perpendicular to flange faces.

Bend

Two flanges are connected by a bend. A bend consists of two coaxial cylindrical faces, which are always tangent to the adjacent planar faces of the flanges.

Bend Relief

Group of faces representing a technological cut between two flanges of different width connected with a bend. The bend relief feature maintains the distance between two opposite faces of the cut.



Corner Relief

Group of faces representing a cut in the corner where three flanges meet together. Corner relief feature maintains the form and size of this cut.

Junction

Junction feature consists of two thickness faces of adjacent flanges, which are not connected via bend.

K-Factor

The material deformation properties for bending is based on the assumption that an internal surface is not deformed when the sheet is bent.

K-factor is a ratio of location of the neutral internal surface to the material thickness as defined by t/T where t = location of the neutral surface and T = material thickness. The K-Factor formulation does not take the forming stresses into account but is simply a geometric calculation of the location of the neutral surface after the forces are applied and is thus the roll-up of all the unknown (error) factors for a given setup. The K-factor depends on many factors including the material, the type of bending operation, the tools, etc. and typically lies between 0.3 to 0.5. The default value of K-Factor for bend radius equal to thickness is 0.27324.

In BricsCAD K-Factor is defined for bend radius equal to the material thickness; to compute the value of K-Factor for bend radius greater than the thickness, BricsCAD uses special interpolation technique proved to be valid for industrial sheet metal applications.

For each sheet metal part, you can specify your own K-factor or use the default one.

To change the K-Factor for your sheet metal part:

4. Select the root node in the Mechanical Browser.
5. Type a value in the K-Factor field.

Bend Table

Bend tables are a more reliable way to express material deformation properties. When you bend a flat sheet of a particular material, you can measure its length before and after this process. You repeat this procedure for different bend angles, bend radius, and sheet thickness and save the measurements in a bend table. Then these measurements can be taken into account to compute the correct unfolded length for all parts made of the same material.

In a bend table, you can add the results for as many different bend angles as you want. However, in most cases it suffices to measure the length for a 90 degrees angle. BricsCAD will

automatically compute the length for other bend angles using a reliable interpolation technique.

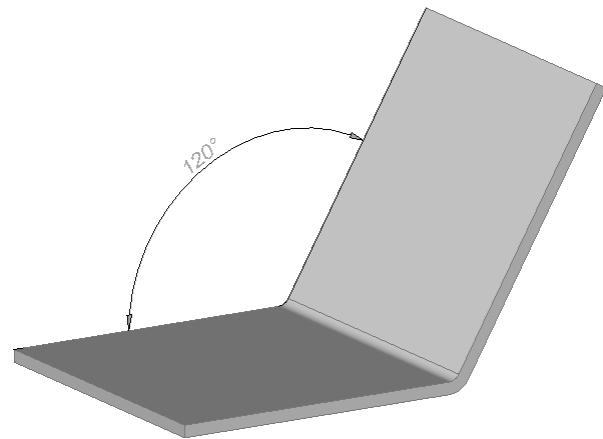
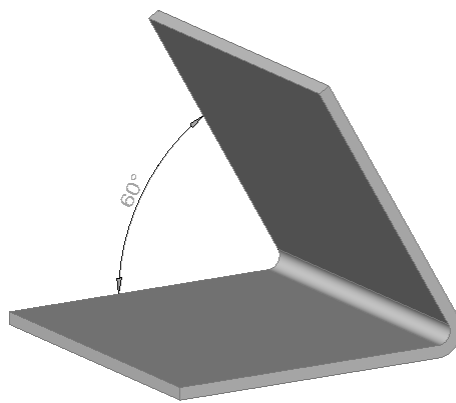
Bend tables are stored in a *.csv (comma separated values) file format. You can create such a table in an ASCII text editor, such as Notepad, but a more reliable and easier way is to create a table in a spreadsheet, such as Microsoft Excel or LibreOffice and then export it to .csv file format.

Bend tables in BricsCAD have the following structure:

	A	B	C	D	E	F	G
1	BricsCAD						
2	Version	1					
3	AngleType	Internal					
4	LengthType	BendDeductionTangent					
5							
6	Thickness	1					
7	AngleValues		45	60	90	120	135
8	Radius	1	1	1.333	2	0.976	0.657
9	Radius	2	0.466	1.081	2.31	1.004	0.641
10	Radius	3	-0.069	0.828	2.621	1.033	0.624
11							
12	Thickness	1					
13	DieWidth	10					
14	AngleValues		45	60	90	120	135
15	Radius	1	1.225	1.533	2.15	1.076	0.732
16	Radius	2	0.616	1.214	2.41	1.071	0.691
17	Radius	3	0.006	0.895	2.671	1.066	0.649

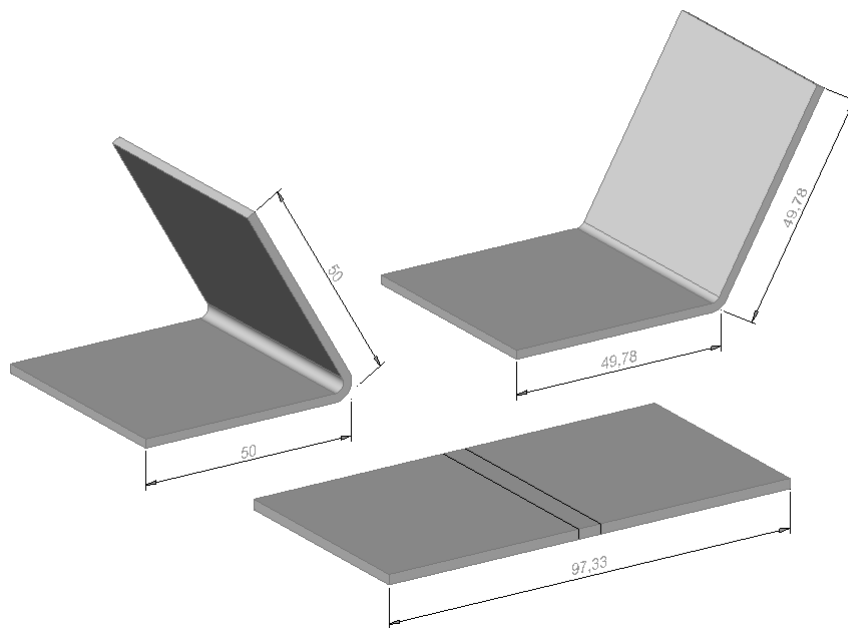
The following rules apply:

- The content of the first cell reads "BricsCAD".
- **Version:** currently supported version is 1.
- **AngleType:** Currently only internal bend angles are supported:



- **LengthType** – the semantics of the value in the cells of the bend table corresponding to a particular bend angle and bend radius. Now only bend deduction measured from the tangent point is supported ("BendDeductionTangent"). Bend deduction is the difference between the sum of the lengths of two flanges measured to the tangent point on a 3D model and the

length of the same fragment in the unfolded state:



- If you produce your sheet metal parts with different tools, you can describe them in one bend table using *DieWidth* parameter. A sample bend table is provided for your convenience in the `..\Samples\Mechanical\bend_tables` subfolder of the BricsCAD program folder (e.g. `C:\Program Files\Bricsys\BricsCAD V14 en_US\Samples\Mechanical\bend_tables`). Please notice that writing rights to this folder might be limited. In such case you need to copy the bend tables to a different folder in which you have full access rights.

To set the bend table for your sheet metal part:

1. Select the root node in the *Mechanical Browser*.
2. Select the *Bend table* field.



3. Click the Browse button (⋮).
4. Select the *.csv file then double click or press the *Open* button.

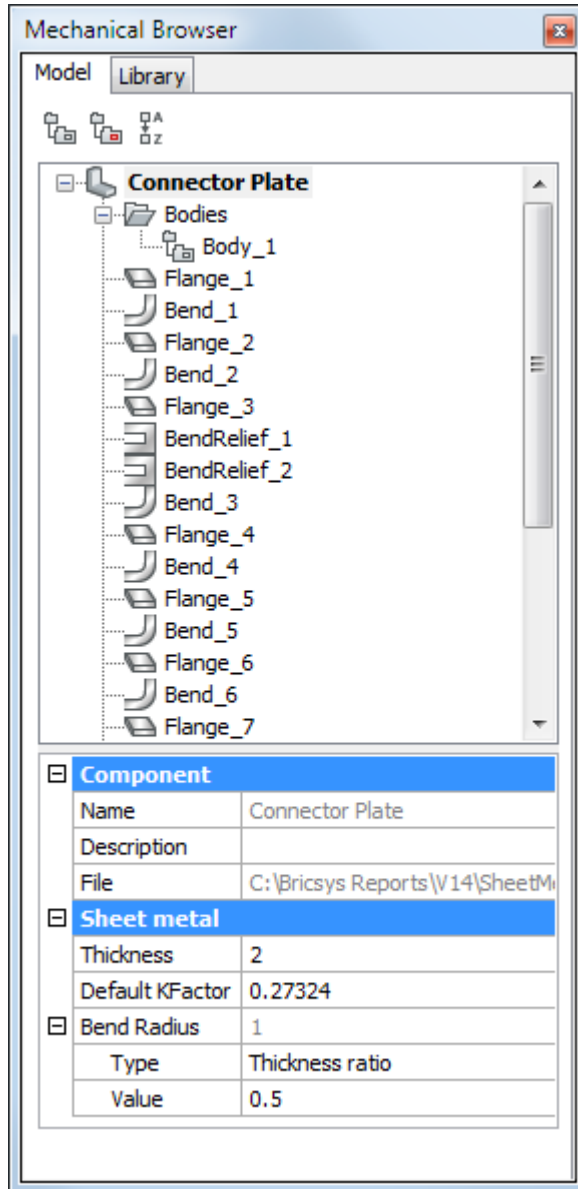
To update the bend table set previously:

5. Select the root node in the *Mechanical Browser*.
6. Select the *Bend table* field.
7. Press the *Update* button (↻).

The Mechanical Browser for Sheet Metal

All features of a sheet metal part are listed in the *Mechanical Browser*.

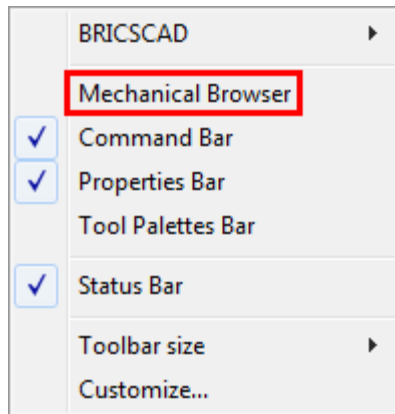
When you select a feature in the *Mechanical Browser*, its faces are highlighted in the model area.



The *Mechanical Browser* automatically opens when switching to the *3D Modeling* workspace.

To open the *Mechanical Browser* manually do the following:

1. Move the cursor over a toolbar, then right click.
A context menu displays:



2. Choose *Mechanical Browser* in the context menu.

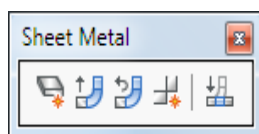
NOTE When you right click a feature in the browser, then select *Dissolve* from the context menu, the selected feature is removed from the part, but it will keep its geometry. However, design intent (spatial and parametric relationships between the feature's faces) associated with the geometry of a dissolved feature is removed.

Commands and Toolbars

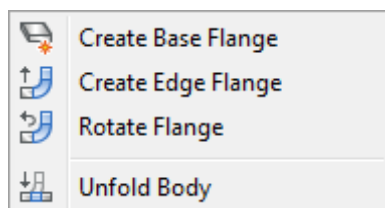
Icon	Command	Description
	SmFlangeBase	Creates a base flange.
	SmFlangeEdge	Creates an edge flange
	SmFlangeRotate	Rotates a flange
	SmUnfold	Unfolds the sheet metal body

The Sheet Metal tools are available:

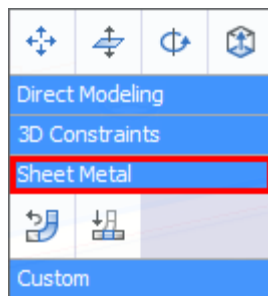
- On the *Sheet Metal* toolbar:



- In the *Sheet Metal* menu:



- In the Quad cursor menu (depending on your selection):




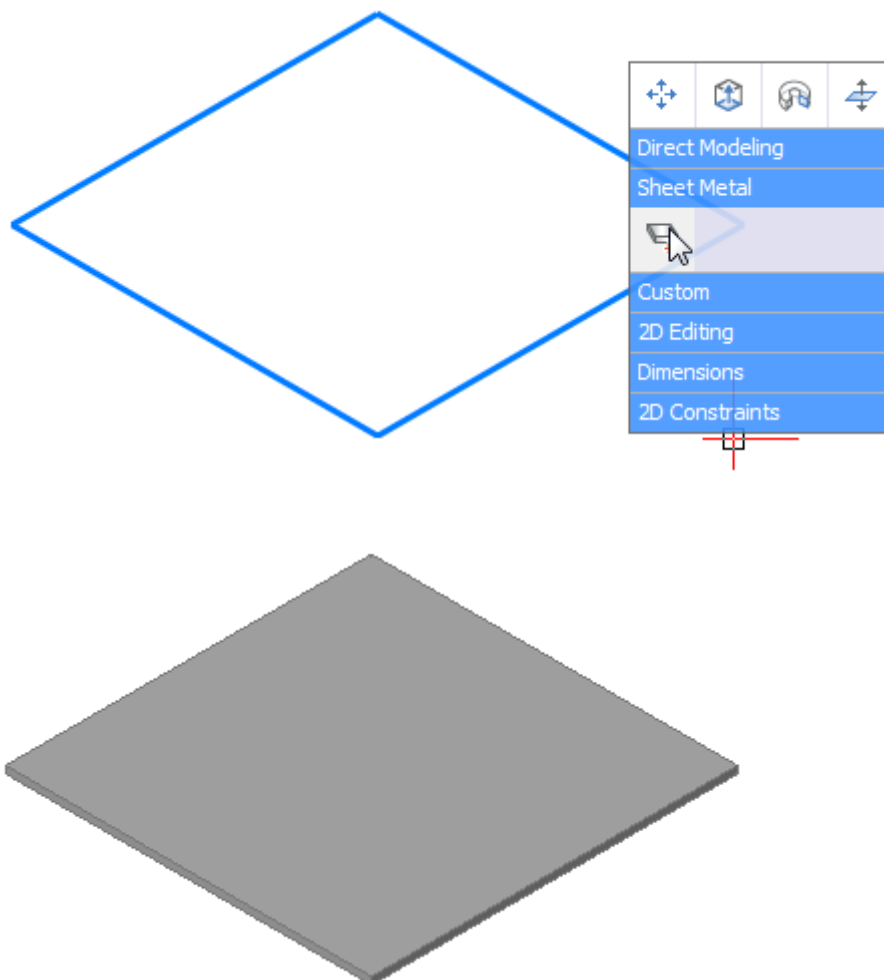
Sheet Modeling Operations

Creating the Base Flange

Start to design your sheet metal part with creating its base flange. To create a base flange, select a closed planar profile and call SmFlangeBase command.

Do one of the following:

- Click the *Create Base Flange* tool button () on the *Sheet Metal* toolbar.
- Choose *Create Base Flange* in the *Sheet Metal* menu.
- Choose *Create Base Flange* in the *Sheet Metal* group of the Quad cursor menu:



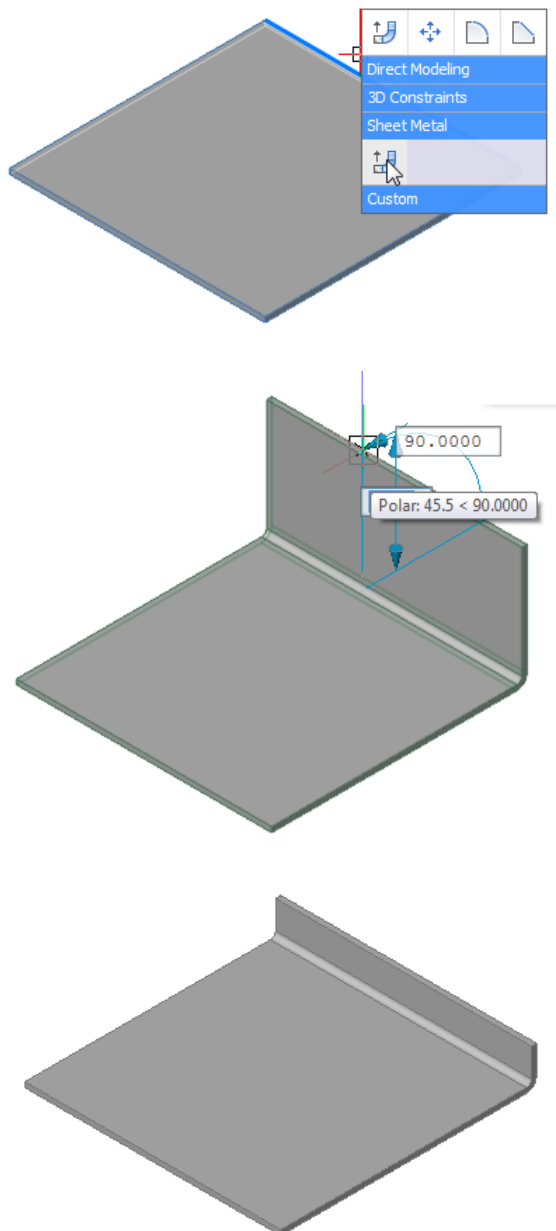
A base flange is a body created by extruding the selected profile to a height equal to the default value of the *Thickness* property of the sheet metal part.

To change the thickness of your sheet metal part, type the appropriate value in the *Thickness* field in the *Mechanical Browser*.

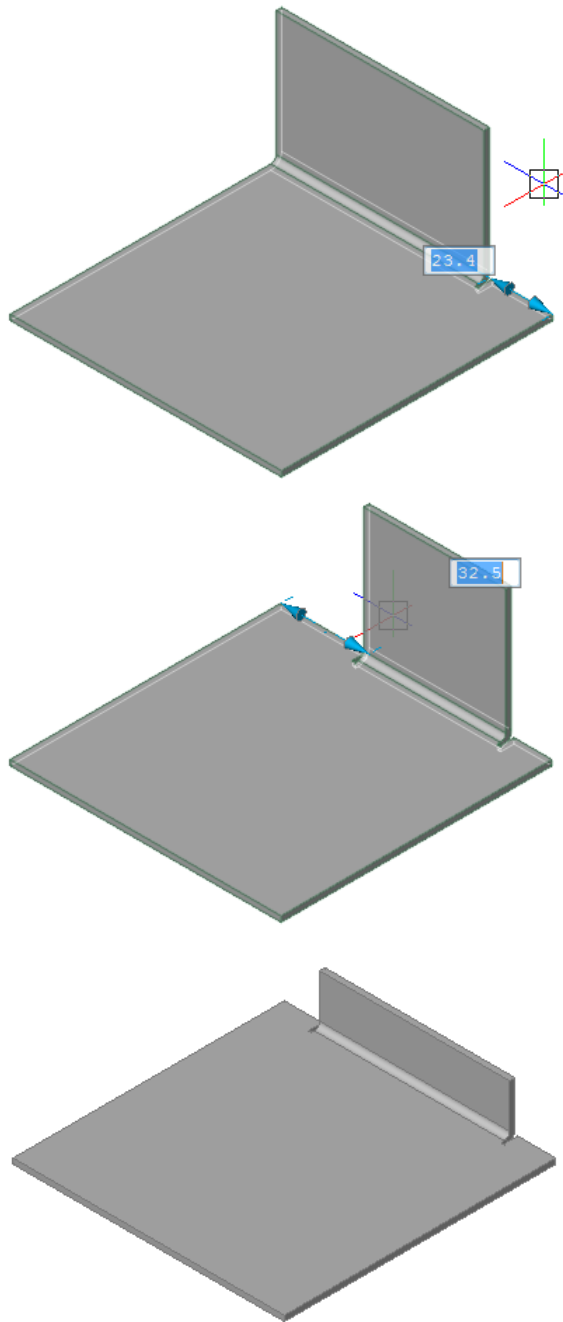
Creating Edge Flanges

You can add an additional flange to your sheet metal part by pulling a linear edge of an existing flange. Select an edge and select the SmEdgeFlange command in the *Sheet Metal* section of the Quad cursor menu.

Move your mouse pointer to define the desired length of the edge flange and the angle between two flanges. You can also use the corresponding dynamic dimensions to type the desired values.



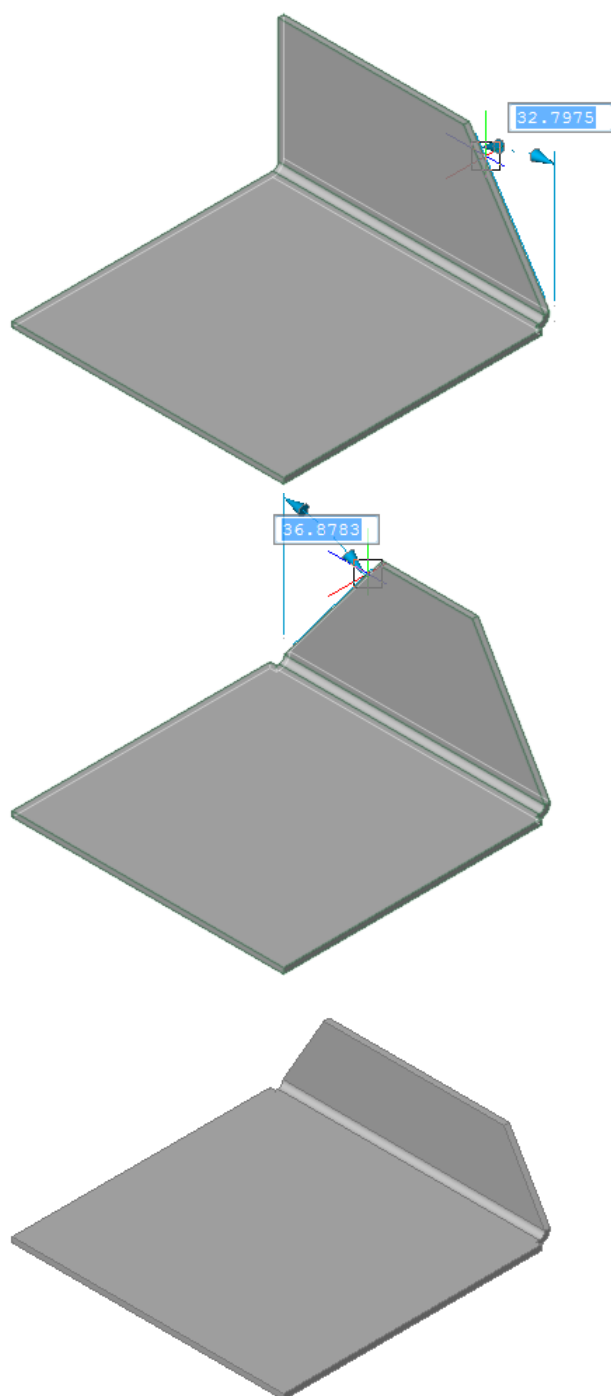
To create an edge flange of which the width is different from the width of the edge, select the *Width* option of SmEdgeFlange command and define offsets from both sides of the edge.



Note that when you create an edge flange of different width, the corresponding bend reliefs are created automatically.

Create Edge Flange With Taper Angle(s)

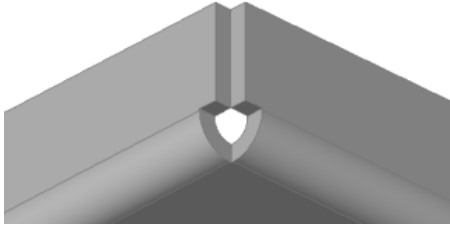
You can create trapezoidal edge flanges using *Taper angle* option of SmEdgeFlange command:



Corner Reliefs and Junctions

When you pull an edge of a flange that is adjacent to a bend edge to create a new flange with SmFlangeEdge command, a corner relief is created automatically in the corner where three flanges meet together:

This operation also creates a junction between two flanges, which are not connected with a bend.



To Rotate a Flange

You can rotate a flange with DmRotate command (⌂), however, you will usually obtain a better result with SmFlangeRotate (⌂). This command automatically selects the rotation axis to respect the design intent of your sheet metal part.

1. Do one of the following:

- Click the *Rotate Flange* tool button (⌂) on the *Sheet Metal* toolbar.
- Choose *Rotate Flange* in the *Sheet Metal* menu.
- Type *smflangerotate* in the command bar, then press Enter.

The command bar reads: Select a flange face to rotate:

Flange faces highlight under the cursor.

2. Click to select a flange face.
The flange rotates dynamically.
A dynamic dimension shows the current angle with respect to the base flange.
3. Do one of the following:
 - Click a point.
 - Type a value in the dynamic dimension field.
 - Press the TAB key, then specify the absolute rotation angle in the dynamic dimension field.

To connect flanges

The SmFlangeConnect command closes gaps between two arbitrarily oriented flanges.

1. Do one of the following:

- Click the *Connect Flanges* tool button (⌂) on the *Sheet Metal* toolbar.
- Choose *Connect Flanges* in the *Sheet Metal* menu.
- Type *smflangeconnect* in the command bar, then press Enter.

The command bar reads: Select planar thickness faces of two flanges:

Flange faces highlight under the cursor.

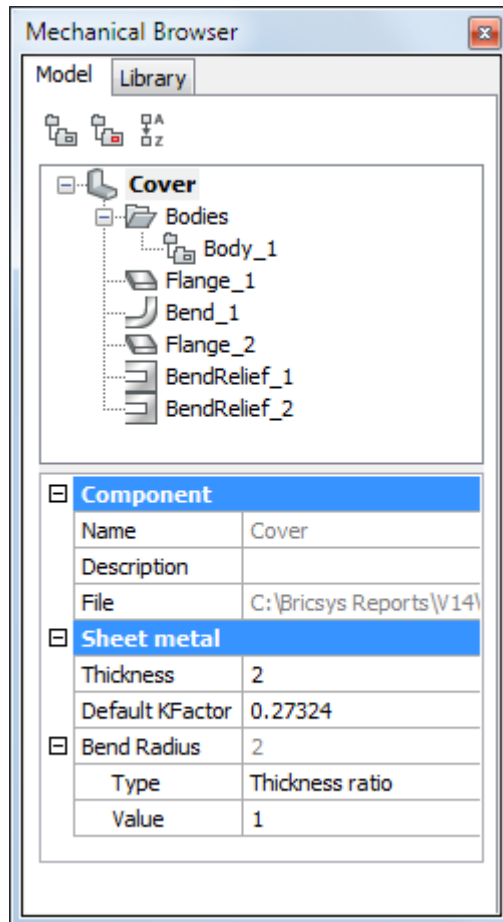
2. Select the thickness face of the first flange.
The command bar reads:
Entities/subentities in set: 1
Select planar thickness faces of two flanges:
3. Select the thickness face of the second flange.
The selected thickness faces are connected.

Changing the Thickness

To change the thickness of a sheet metal part:

1. Select the root node in the *Mechanical Browser*.

2. Type a value in the *Thickness* field.



Changing the bend radius

To change the overall bend radius:

1. Select the root node in the *Mechanical Browser*.
2. Do one of the following:
 - Set *Type* to *Absolute Value*, then type the thickness in the *Value* field. The *Bend Radius* field changes accordingly.
 - Set *Type* to *Thickness ratio*, then type a value in the *Value* field. The *Bend Radius* field is calculated as the product of the *Thickness* and the *Thickness ratio*.

To change the bend radius for a particular bend:

1. Select the bend node in the *Mechanical Browser*. The selected bend highlights in the model.
2. Do one of the following:
 - Set the *Type* to *Global value*. The current global bend radius is applied to the selected bend.
 - Set the *Type* to *Absolute value*, then type the thickness in the *Value* field. The *Bend Radius* field changes accordingly.
 - Set the *Type* to *Thickness Ratio* and type a value in the value field. The *Bend Radius* field is calculated as the product of the *Thickness* and the *Thickness ratio*.

To Unfold the Sheet Metal Body

To unfold your sheet metal body, select a start flange face and run SmUnfold command . BricsCAD will automatically create a solid body corresponding to a flat sheet of metal needed to manufacture your sheet metal part using bending techniques. This sheet is placed on XY-plane and is oriented along the coordinate axes similarly to the orientation of the initial body in 3D space. To change the orientation of the unfolding in OZ axis, run SmUnfold command again and select the opposite face of the initial body .

The SmUnfold command prompts you to either:

Keep the 3D solid in the model.

- Save the 3D solid in a separate drawing file.
- Convert the solid to a 2D drawing file (*.dwg or *.dxf)
- Overall dimension and bending annotations for manufacturing are added to the exported drawing automatically.

The SmUnfold command takes the deformation of the sheet metal material during bending into account. When a flat sheet of metal is bent into a 3D part with a bending tool (like a press break), the material is plastically deformed – it is compressed inside the bend and stretched outside of it. So the length of the part measured along its surface is different in flat and bend states. BricsCAD is able to automatically compute the proper unfolded length of your part based on the material deformation properties. These properties can be defined by setting the value of K-Factor parameter or by attaching a bend table.

Direct Modeling Overview

BricsCAD provides advanced easy-to-use functions for working with ACIS solid geometry, including:

- The creation of solid geometry using Extrude (dmExtrude) and Revolve (dmRevolve) commands applied to 2D contours
- The creation of fillets (dmFillet) and chamfers (dmChamfer) on existing solid geometry.
- Modifying (dmPushPull) or deletion (dmDelete) of fillets and chamfers.
- Direct interactive manipulation of solid faces using PushPull (dmPushPull), Move (dmMove) and Rotate (dmRotate) operations.
- The creation or removing of a volume when 2D contour attached to a face of a solid is being pushed or pulled.

All these operations are performed in interactive mode: the intermediate result of the operation is shown dynamically, according to user input defined by mouse movement.

The key feature of direct editing capabilities provided by BricsCAD consists in preserving of design intent. Design intent can be expressed by 3D dimensional and geometrical constraints which can be explicitly specified by user (see 3D Constraints). 3D dimensional constraints (Distance, Radius and Angle) imposed for faces and edges give user full parametric control over the solid model, different configurations and scales of the model can be obtained simply by changing the parameter of a dimensional constraint. Geometrical constraints (Coincident, Tangent, Parallel and others) define how these changes influence the geometry of the model. However, there is no need to introduce all the geometrical constraints manually since most of them can be recognized automatically by BricsCAD (see Design Intent) to provide the expected result.

Another feature of Direct Editing operations is that they can be easily applied with the Quad cursor menu. This tool provides a set of operations for the currently selected entities. At the same time the Quad provides the other basic solid editing operations such as copy, boolean union and subtract.

Direct modeling operations apply to all ACIS solid geometry, whether created in BricsCAD or imported, and can be combined with regular solid modeling operations, for example SOLIDEDIT.

3D Constraints

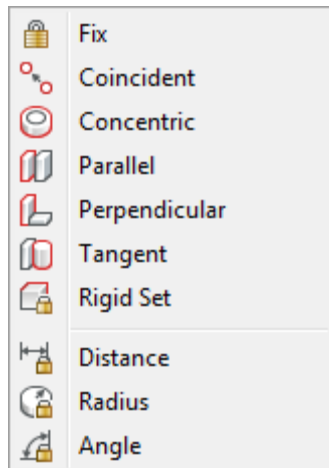
Commands: DMCONSTRAINTSBAR, DMUPDATE, DMFIX3D, DMCOINCIDENT3D, DMCONCENTRIC3D, DMPARALLEL3D, DMPERPENDICULAR3D, DMTANGENT3D, DMDISTANCE3D, DMRADIUS3D, DMANGLE3D

Variational Direct Modeling

BricsCAD offers powerful tools for parametric direct solid modeling using geometric constraints solving (aka variational direct modeling). Solid models can be modified by defining constraints between its elements (edges and faces), as soon as constraint is added BricsCAD automatically modifies the model accordingly, and these constraints are kept satisfied upon following modifications of the model. The key advantage of variational direct modeling is that all constraints are taken into account simultaneously and the model behavior does not depend on the constraints creation order. It allows you to parameterize any feature of the 3D model and to do not care about the model creation history.

Dimensional constraints allow controlling the dimensions of the model, when such constraint is introduced or the value of its parameter is changed BricsCAD automatically updates the solid geometry to satisfy the constraint. For example, the dimensions of a box can be controlled by the parameters of three distance constraints applied to its opposite faces.

Constraints can be created using *3D Constraints* toolbar or *Parametric/3D Constraints* menu:



BricsCAD provides the following 3D constraints:

- Geometrical constraints: *Fix* (🔒), *Coincident* (🔗), *Concentric* (🔗), *Parallel* (||), *Perpendicular* (⊥), *Tangent* (🔗) and *Rigid Set* (🔗).
- Dimensional 3D constraints: *Distance* (📏), *Radius* (⊙), *Angle* (📐).

Mechanical Browser

The *Mechanical Browser* (🔍) allows to navigate through all the constraints in the model and to edit dimensional constraints.

See the BmBrowser command to learn more about the *Mechanical Browser*.

3D Constraints types

The following entities are supported for constraints creation:

- Faces of ACIS solids of planar, cylindrical, spherical, toroidal and conical geometry.
- Linear and circular edges of ACIS solids

3D Constraints and Direct Modeling

3D Constraints are taken into account when direct modeling operations are applied. If there is a 3D constraint which fixes the placement of faces or edges this constraint will be preserved during Push/Pull, Move and Rotate operations. The dynamically presented result of the operation respects existing constraints as well.

If some faces or edges disappear in a direct modeling or in a Boolean solid editing operation 3D constraints applied to such entities are removed. However, if there is an exact correspondence between the initial faces and the faces obtained in a result of a Boolean operation 3D constraints are automatically applied to new entities.

If 3D constraints prevent modification of the model with a particular direct modeling operation (e.g. a Push/Pull operation applied to a face with a *Fix* constraint) the status line of the 3D Constraints Bar receives a *No Solution* status.

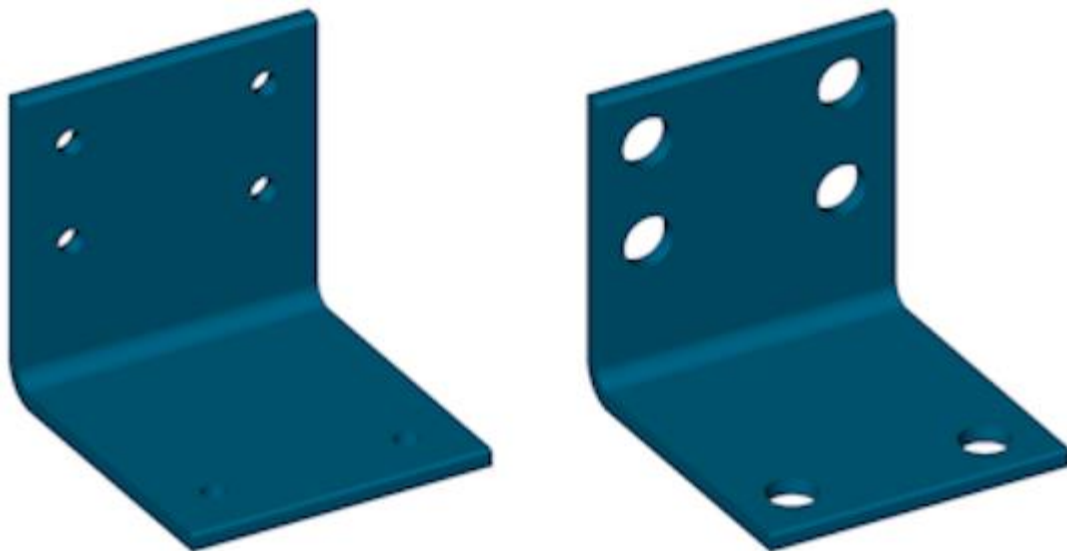
Depending on the value of the DMRECOGNIZE system variable, BricsCAD automatically recognizes geometrical relations between surfaces of a solid and preserves them during direct modeling operations (see Design Intent Recognition).

Design intent recognition



Design Intent recognition (Internet connection needed)

Direct modeling operations allow to change geometry of 3D solid models easily: when a single face of a solid is modified with a particular operation other faces are modified accordingly in order to keep design intent of the model and its topological and geometrical validity. For example, consider a model with a number of holes of the same radius. When the radius of a cylindrical face is modified with a push/pull operation or by changing the value of a radial constraint other faces that share the same diameter are recognized by BricsCAD and modified automatically to keep the radii synchronized.



Apart from the recognition of equal radii, BricsCAD automatically recognizes other geometrical relations between surfaces of a solid and preserves them during direct modeling operations.

To define the design intent recognition, click the buttons on the *Design Intent* toolbar. A pressed button indicates the feature is selected. An unpressed button indicates the feature is not active.



- tangent surfaces (planes, cylinders and cones)
- coincident planes
- parallel planes
- perpendicular planes
- cylinders perpendicular to planes
- coaxial surfaces (cylinders and cones)
- surfaces of equal radius (cylinders and spheres)
- switch off/on all

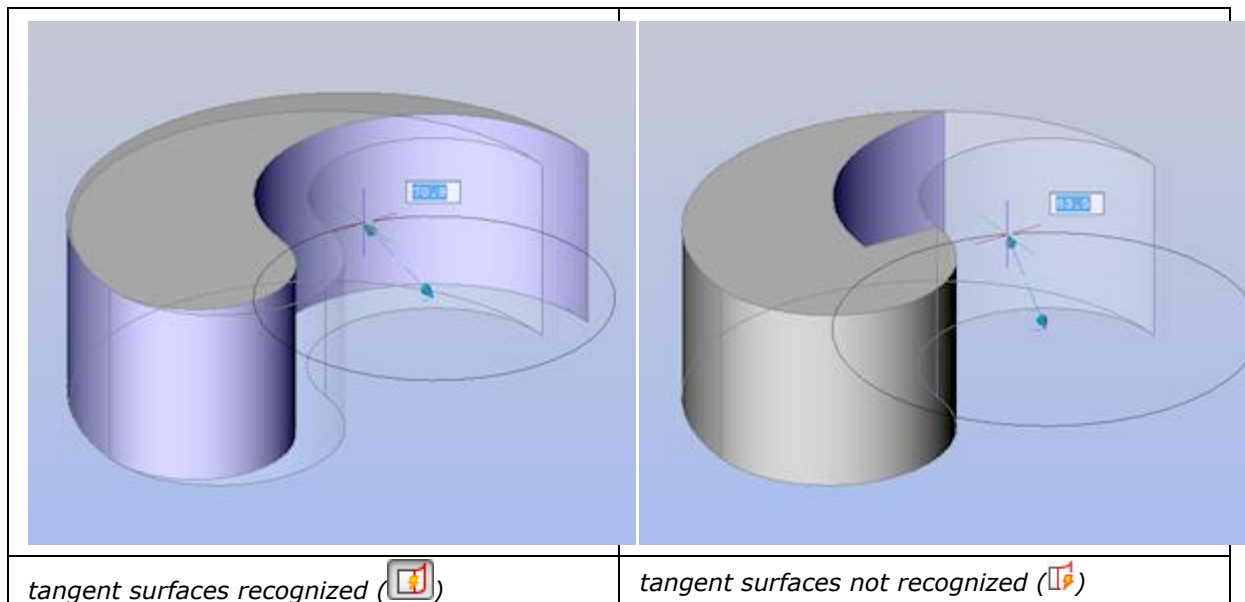
The design intent options can also be set by editing the DMRECOGNIZE system variable in the Settings dialog.

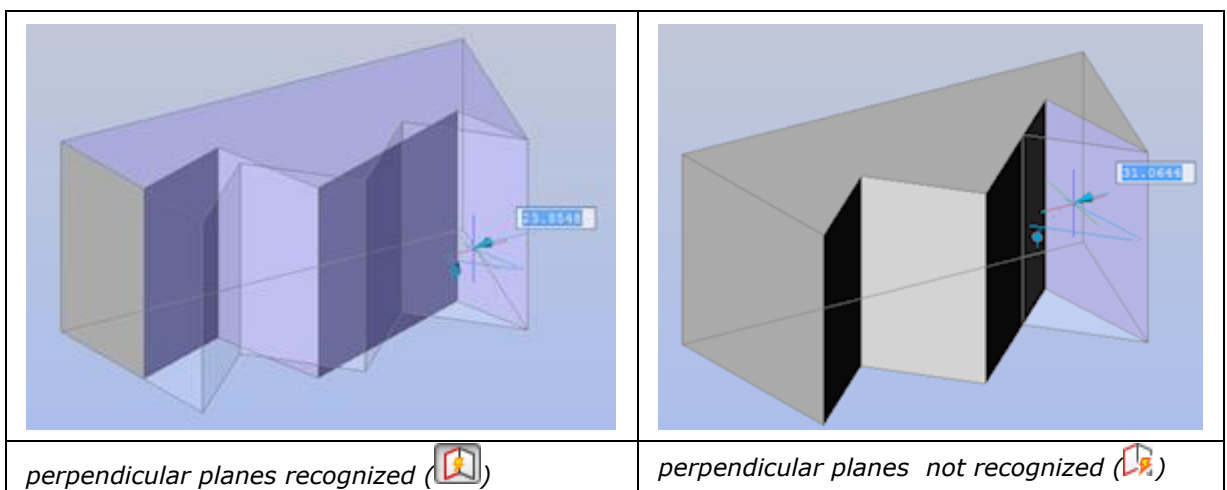
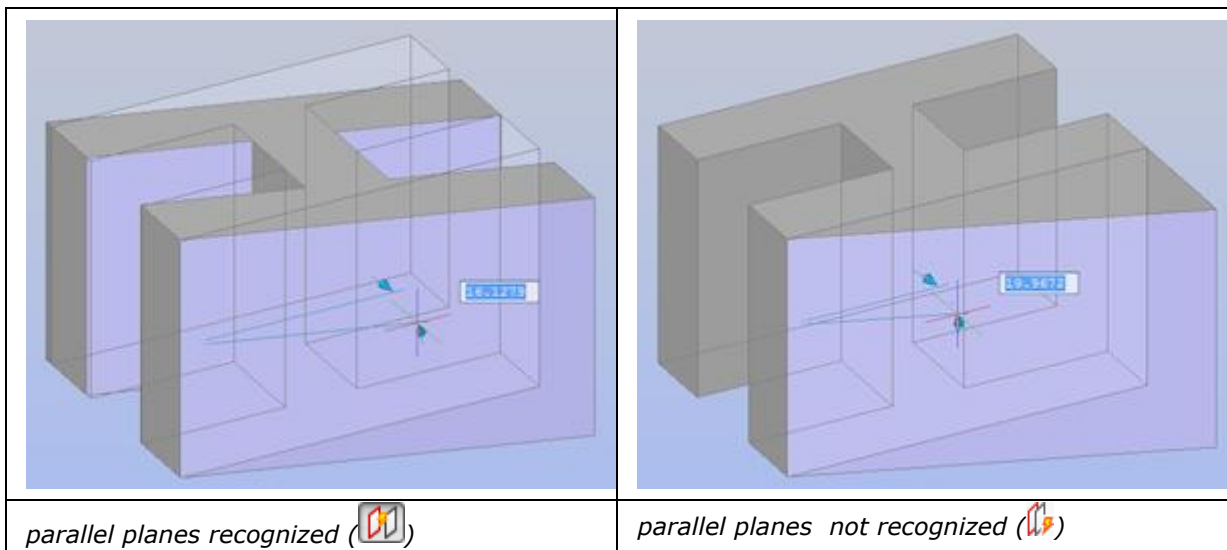
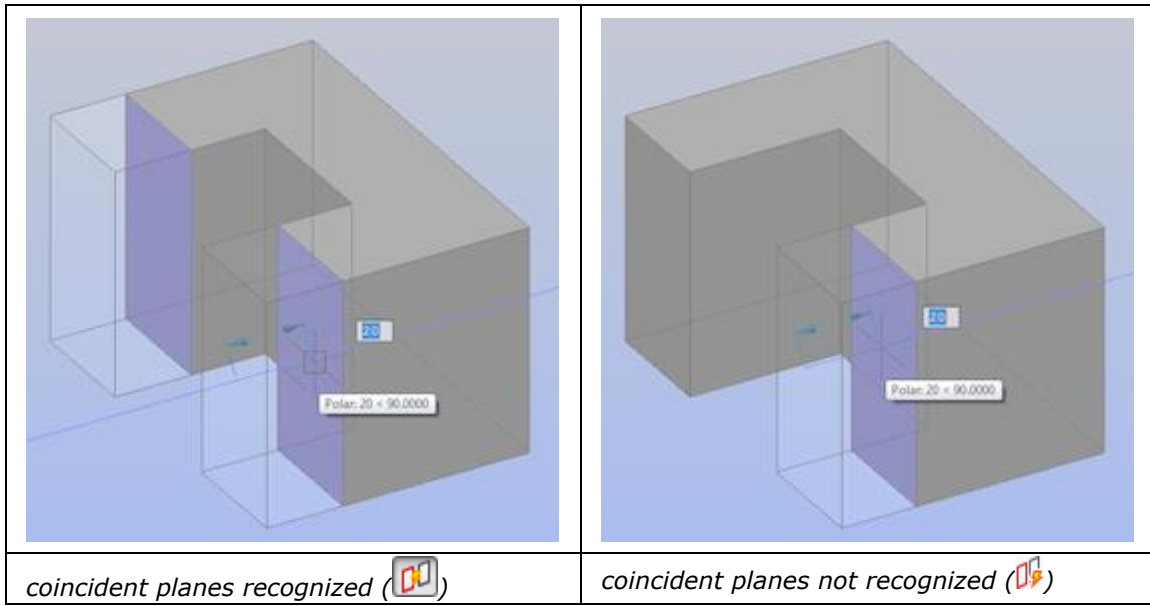
Click the *Options* tool button on the *3D Constraints* toolbar to set the *Automatic 3D geometry constraints recognition*. The state of tool buttons on the *Design Intent* toolbar (pressed or unpressed) is adjusted accordingly.

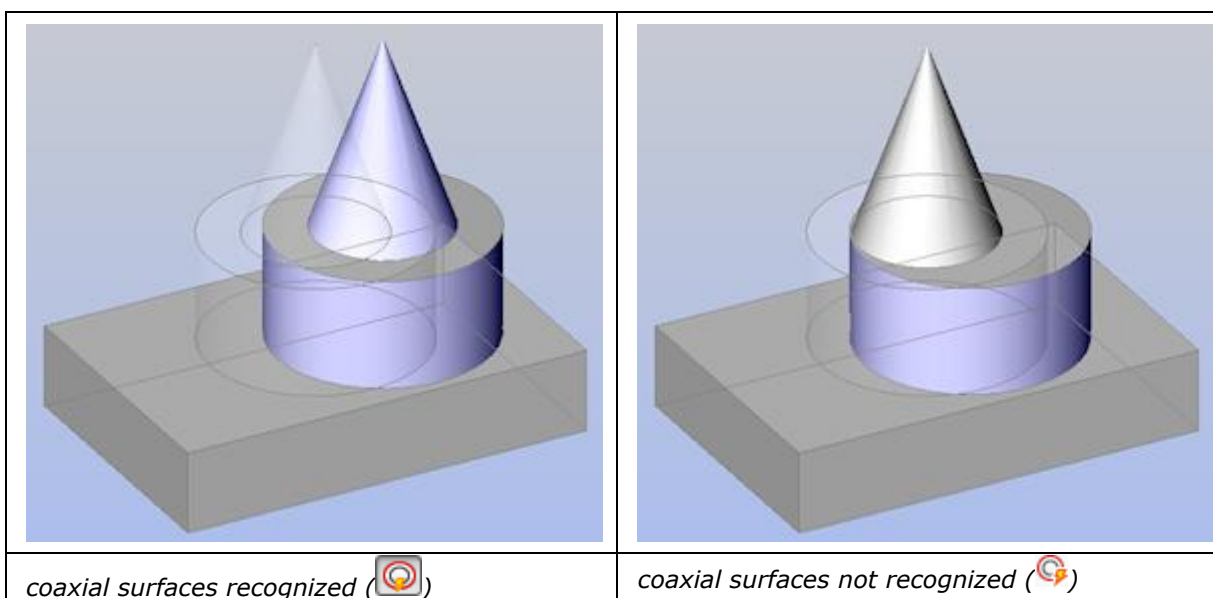
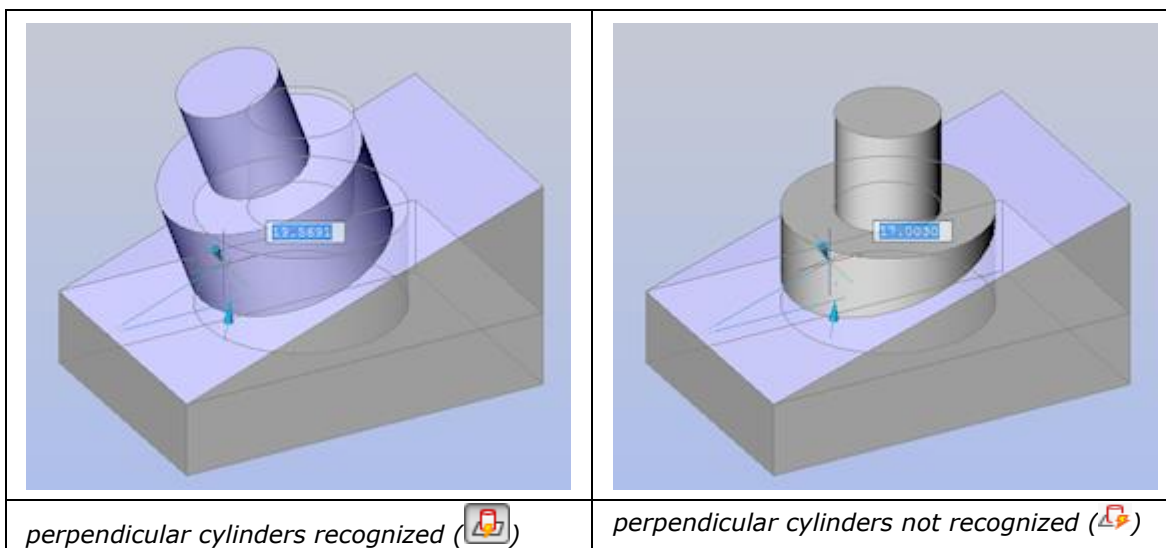
<input checked="" type="checkbox"/> Automatic 3D geometry constraints recognition	0x0066 (102)
negative	<input type="checkbox"/> Switch off automatic 3D geometry constraints recognition
0x0001	<input type="checkbox"/> Tangent surfaces
0x0002	<input checked="" type="checkbox"/> Coincident planes
0x0004	<input checked="" type="checkbox"/> Parallel planes
0x0008	<input type="checkbox"/> Perpendicular planes
0x0010	<input type="checkbox"/> Cylinders perpendicular to planes
0x0020	<input checked="" type="checkbox"/> Coaxial surfaces
0x0040	<input checked="" type="checkbox"/> Cylinders and spheres of equal radius

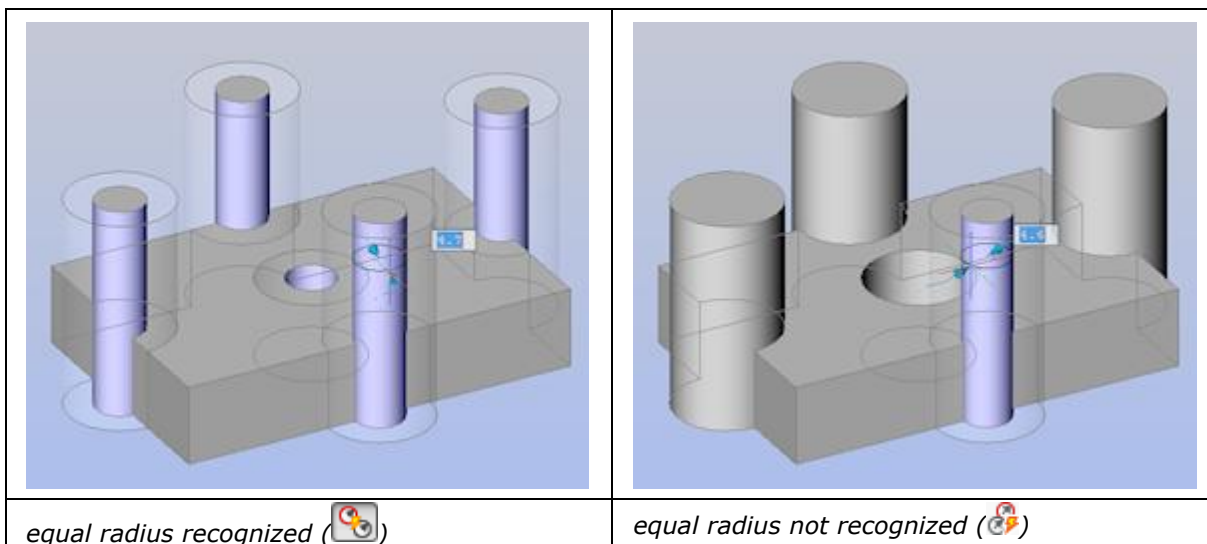
NOTE 3D Constraints which are explicitly defined by the user are taken into account when solid model is being modified with direct modeling commands. These constraints have a higher priority than automatically recognized ones and allow the user to better control the behavior of solid model modifications.

The following examples show the difference in the behavior whether automatic 3D geometry constraints are recognized or not.





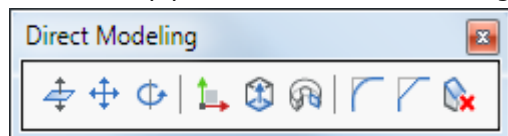




Direct Modeling operations

This section describes direct modeling operations offered by BricsCAD. These operations are available from the Quad tool.



Alternatively you can use *Direct Modeling* Menu or *Direct Modeling* Toolbar:









The *Align UCS* tool () launches the UCS command with the *Face* option selected.

If the UCSDETECT system variable is *ON*, the ucs is aligned automatically to the currently highlighted face. The UCS-icon changes accordingly. Press the TAB key to select an obscured face. Click the *DUCS* field in the Status Bar to toggle the dynamic ucs feature.

The following direct modeling operations are available in BricsCAD:

Icon	Command	Applies to	Description
	dmPushPull	Planar, cylindrical, spherical, conical and toroidal faces of a solid. Closed 2D entities, attached to a solid.	Faces: adds volume to, or removes volume from the solid by cursor movement or direct distance input. Closed 2D entity (If attached to a face of a solid): creates a solid by extruding the 2D entity. The height of the extrusion is specified by cursor movement or direct distance input. Depending on the extrusion direction the newly created solid is unified with or subtracted from the existing solid the 2D entity was attached to.
	dmMove	Cylindrical, spherical, conical and toroidal faces of a solid. Linear and circular edges of a solid. Solids.	Moves the selected geometry using a vector in the XY- plane of the current UCS. When a face or an edge of a solid is moved, adjacent faces and edges are adjusted to preserve the correct solid topology.

	dmRotate	Planar, cylindrical, spherical, conical and toroidal faces of a solid. Solids.	Rotates the selected geometry around an axis. When a face of a solid is rotated, adjacent faces and edges are adjusted to preserve the correct solid topology.
	dmFillet	Sharp edges of a solid.	Creates a fillet between adjacent faces sharing a sharp edge(s).
	dmChamfer	Sharp edges of a solid.	Creates a chamfer between adjacent faces sharing a sharp edge(s).
	dmExtrude	Closed 2D entities.	Creates solids by extruding the selected 2D entities.
	dmRevolve	Closed 2D entities.	Creates a solid by revolving a 2D entity about an axis.
	dmDelete	Faces of a solid. Closed 2D entities. Solids.	Deletes the selected entities. When faces of a solid object are deleted, the gap is filled by extending the adjacent faces.

Selecting geometry

- Press and hold the *TAB* key to select obscured geometry.
- The *PREVIEWTOPDOWN* system variable controls whether individual faces of 3D solids highlight first or the entire 3D-solid.
When the Shift and Ctrl keys are pressed simultaneously and not held down longer than 500 milliseconds, they toggle the *PREVIEWTOPDOWN* setting.
- Shift-space-click cycles the preview of faces, edges and entities (in that order).
- If a direct modeling command is entered at the command prompt or selected in a menu or toolbar, you need to press the Enter key (or right click) to conclude the selection procedure.
- If launched from the Quad cursor menu, the command is executed when you click to confirm the selection of the currently highlighted geometry; press and hold the Ctrl key to select multiple entities.

To control dimensions

When using the *dmPushPull* command a dynamic dimension displays. Type the desired distance or radius in the dynamic field, then press Enter to confirm.

The initial distance is measured from the previous position of the face. Repeatedly press the TAB key to cycle through the distances to other references planes (if any). Reference planes are parallel to the face being pushpulled. Press and hold the Shift key, then press the TAB key to cycle in reverse order.

Making holes

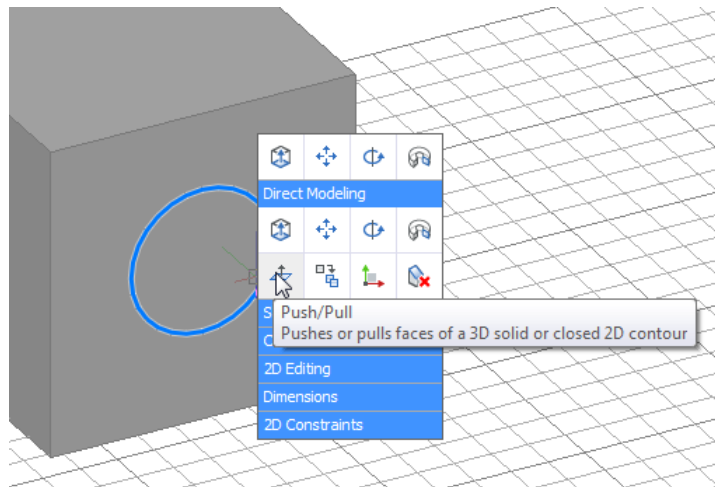
Typically holes are made using the *SUBTRACT* command.

When pushpulling a closed 2D entity that lies on a face of a solid, a hole is created when you push the 2D entity through the solid.

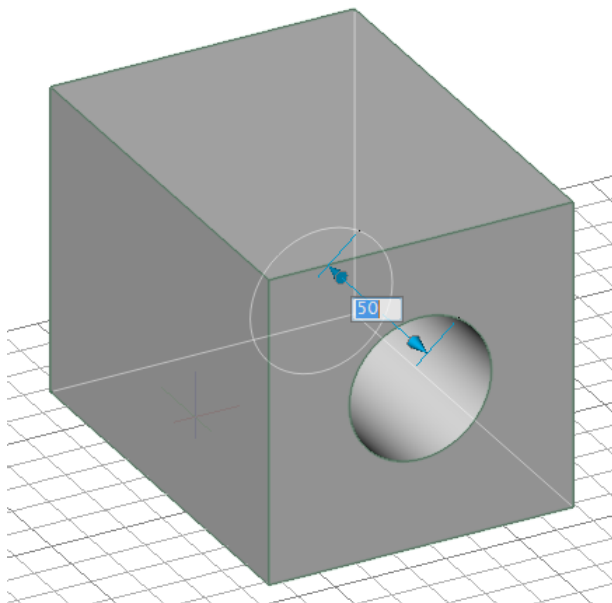
To make a (circular) hole through a solid

1. Make sure Dynamic UCS is active.
2. Launch the Circle command.

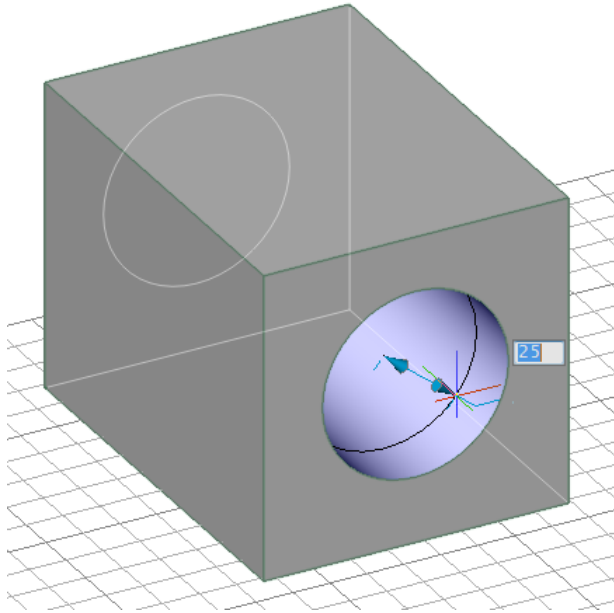
3. Move the cursor to the face of the solid you want to create the circular hole in. The UCS automatically aligns to the face the cursor is on.
4. Draw a circle on the face of the solid.
5. Hover over the circle, then choose *Push/Pull* in the *Direct Modeling* command group of the Quad cursor menu.



6. Push the circle inside the solid.



7. Do one of the following:
 - Type a distance in the dynamic field to define the depth of the hole.
 - Push the circle through the solid to create a through-hole.
8. (option) Choose *Push/Pull* in the *Direct Modeling* command group of the Quad cursor menu to modify the radius of the hole.



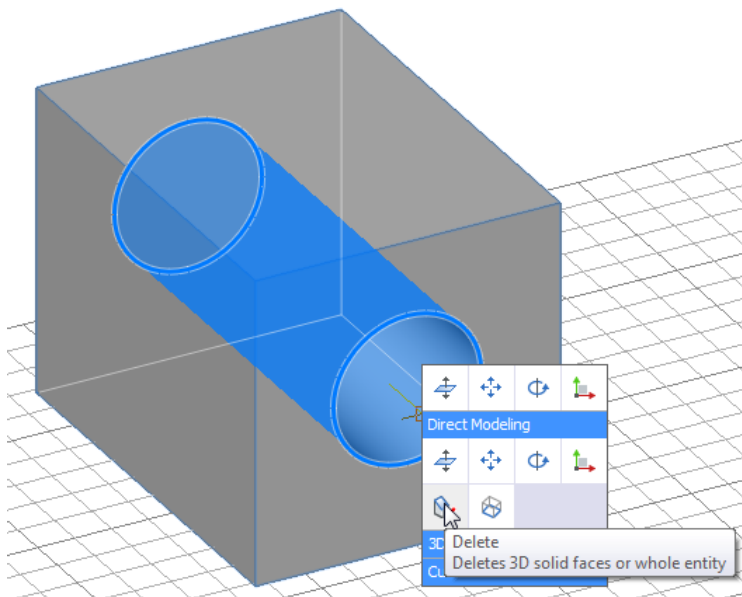
9. (option) Use the dmDistance3d command to apply distance constraints between the side faces of the solids and the center of the cylindrical surface of the hole to control the position of the hole.
10. (option) Use the dmRadius3d command to control the radius of the hole.

Deleting holes.

To delete a hole, you must delete all internal faces and/or surfaces of the hole.

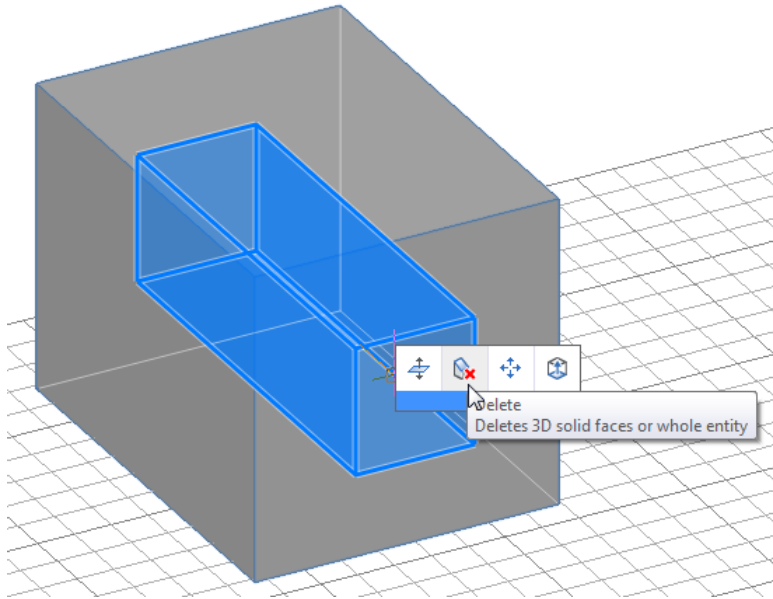
To delete a circular hole

1. Hover over the cylindrical surface of the hole, then click to select the face.
2. Choose *Delete* in the *Direct Modeling* command group of the Quad cursor menu.



To delete a rectangular hole

3. Hover over one of the visible internal faces of the hole.
The face highlights.
4. Click to select the face.
5. Move the cursor to the next internal face of the hole.
Repeatedly press the TAB key until the face highlights, then click to select the face.
6. Repeat the previous steps until all internal faces of the hole are selected.
7. Choose *Delete* in the *Direct Modeling* command group of the Quad cursor menu.



Working with sections

Commands: SECTION, SECTIONPLANE, LIVESECTION, SECTIONPLANESETTINGS, SECTIONPLANETOBLOCK.

The *Section* command creates section planes of 3D solids; the results are region entities.

The *Sectionplane* command creates a section entity that creates sections of 3D solids, surfaces and meshes.


The *Livesection* command toggles the *Live Section* property of a section plane.

The *Sectionplanesettings* command defines the properties of a section plane entity in the *Drawing Explorer - Section Planes* dialog.

The *Sectionplanetoblock* command saves the selected section plane to a 2D cross section / elevation block or a 3D cutaway section block.

To launch the Sectionplane command

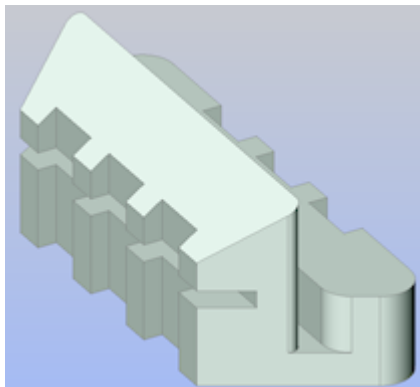
Do one of the following:

- Click the *Section Plane* tool button () on the *Sections* toolbar.
- Choose *Sections > Section Plane* in the *Model* menu.
- Type *sectionplane* in the command bar.

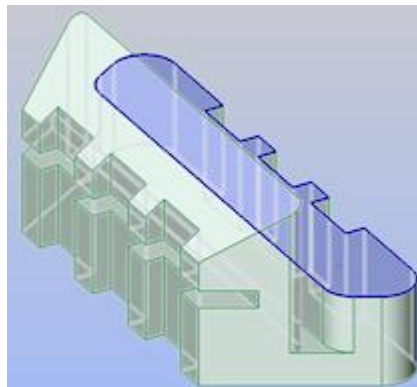
The command bar reads: Specify start point or [select Face/Draw/Orthographic]:

Align a section plane to a face

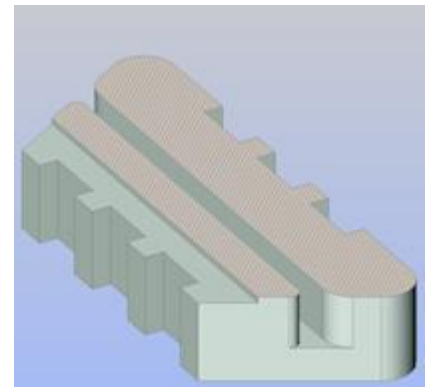
1. Launch the Sectionplane command.
2. Choose *Select Face* in the prompt menu or type *F* in the command bar.
The command bar reads: Select entity:
3. Select the face of a solid you want to align the section plane to.



Solid



Select face



Section plane aligned to face

The section plane is aligned to the selected face. The *Live Section* property of the section entity is switched on automatically.

4. (option) Edit the properties of the section plane.

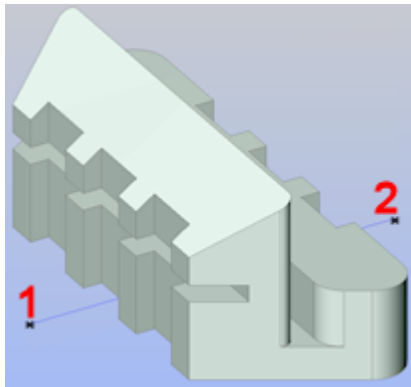
To create a vertical section plane

1. Make sure the WCS is the current coordinate system.
2. Launch the Sectionplane command.
3. Specify two points.
The section plane is created through these points and perpendicular to the XY-plane

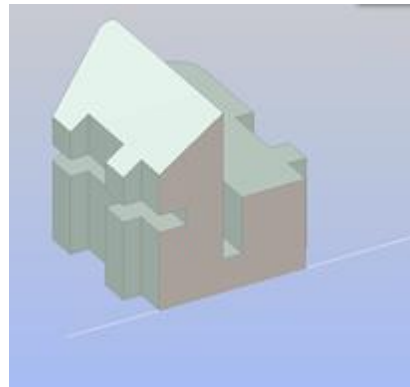
of the current coordinate system.

The part of the solid at the right hand side of the section plane is cut away.

The *Live Section* property of the section entity is not switched on automatically.



Specify two points



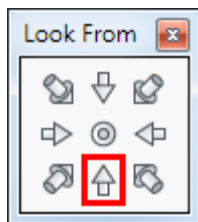
Vertical section plane

(Live section)

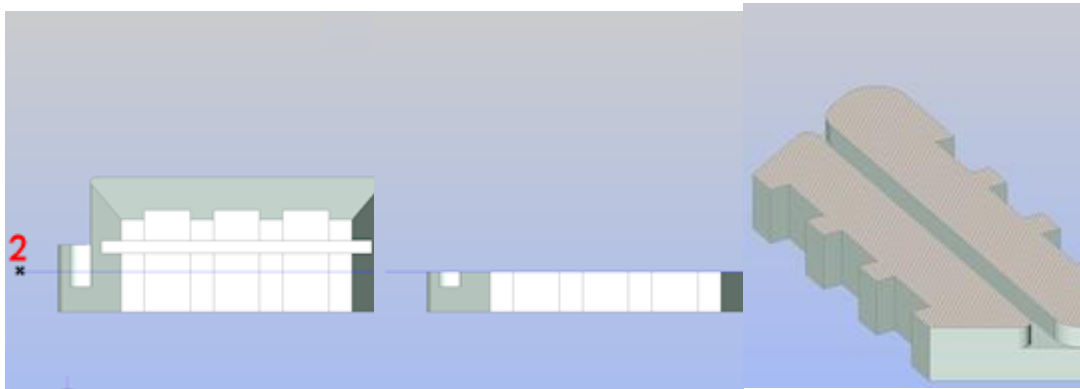
4. (option) Edit the properties of the section plane.

To create a horizontal section plane

1. Choose *Front* in the *Look From* toolbar.



2. Align the UCS to the front view:
Type *ucs* at the command prompt, choose the *View* option in the prompt menu or type *V* in the command bar.
3. Launch the *Sectionplane* command.
4. Specify two points.
The section plane is created through these points and perpendicular to the XY-plane of the current coordinate system.
The part of the solid at the right hand side of the section plane is cut away.
The *Live Section* property of the section entity is not switched on automatically.



Specify two points

The part of the solid at the right hand side of the section plane is cut away.

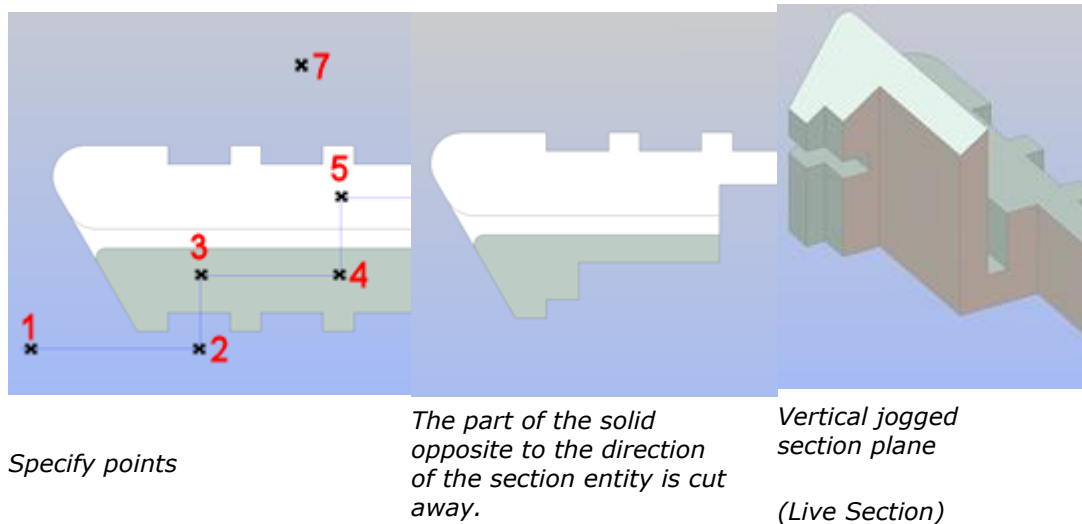
Horizontal section plane

(Live Section)

5. (option) Edit the properties of the section plane.

To create a vertical jogged section plane

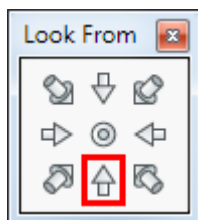
1. Make sure the WCS is the current coordinate system.
2. Launch the Sectionplane command.
3. Choose *Draw* in the prompt menu or type *D* in the command bar.
The command bar reads: Specify start point:
4. Specify the first point (1).
The command bar reads: Specify next point:
5. Specify the second point (2):
The command bar reads: Specify next point or ENTER to finish:
6. Specify the other points (3 - 6) , the right click or press Enter to stop.
The command bar reads: Specify the direction of the section entity:
7. Click a point (7) to specify the direction of the section entity.
The part of the solid opposite to the direction of the section entity is cut away.
The *Live Section* property of the section entity is not switched on automatically.



8. (option) Edit the properties of the section plane.

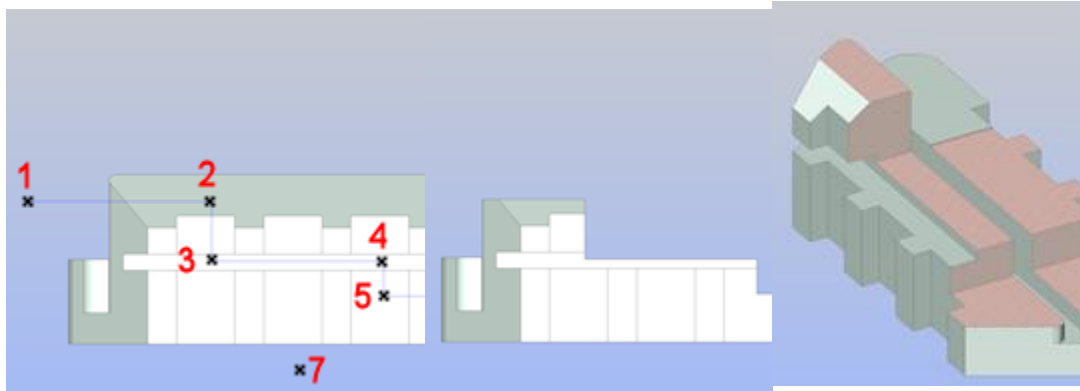
To create a horizontal jogged section plane

1. Choose *Front* in the *Look From* toolbar.



2. Align the UCS to the front view:
Type *ucs* at the command prompt, choose the *View* option in the prompt menu or type *V* in the command bar.
3. Launch the Sectionplane command.
4. Choose *Draw* in the prompt menu or type *D* in the command bar.
The command bar reads: Specify start point:
5. Specify the first point (1).
The command bar reads: Specify next point:

6. Specify the second point (2):
The command bar reads: Specify next point or ENTER to finish:
7. Specify the other points (3 - 6) , the right click or press Enter to stop.
The command bar reads: Specify the direction of the section entity:
8. Click a point (7) to specify the direction of the section entity.
The part of the solid opposite to the direction of the section entity is cut away.
The *Live Section* property of the section entity is not switched on automatically.
The section plane gets the *Boundary* state.



Specify points

The part of the solid opposite to the direction of the section entity is cut away.

Horizontal jogged section plane

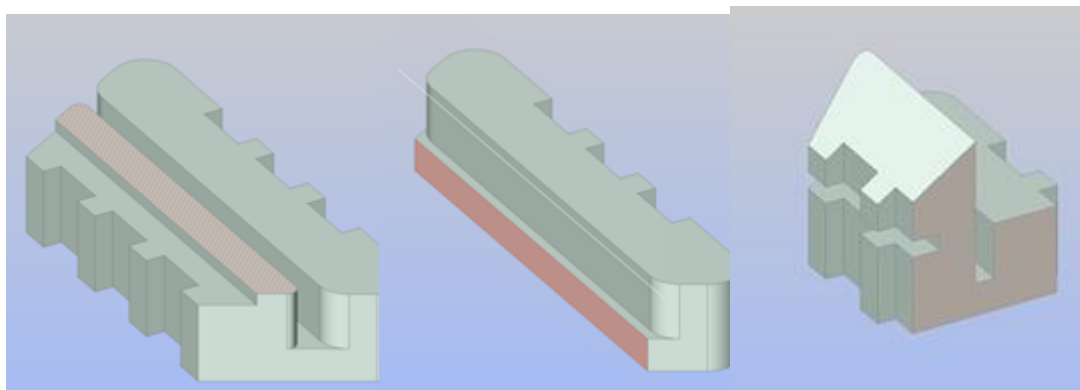
(Live Section)

9. (option) Edit the properties of the section plane.

To create a orthographic section plane

1. Make sure the WCS is the current coordinate system.
2. Launch the Sectionplane command.
3. Choose *Orthographic* in the prompt menu or type *O* in the command bar.
The command bar reads: Align section to [Front/bAck/Top/Bottom/Left/Right]
<Top>:
4. Pick the orthographic section orientation of your choice in the prompt menu or type the corresponding option in the command bar.
Orthographic section planes are created through the center of the solid, parallel to the XY-plane (Top and Bottom), YZ-plane (Left and Right) or XZ-plane (Front and Back) of the current coordinate system.

The *Live Section* property of the section entity is not switched on automatically.



Top

Front

Right

- (option) Edit the properties of the section plane.

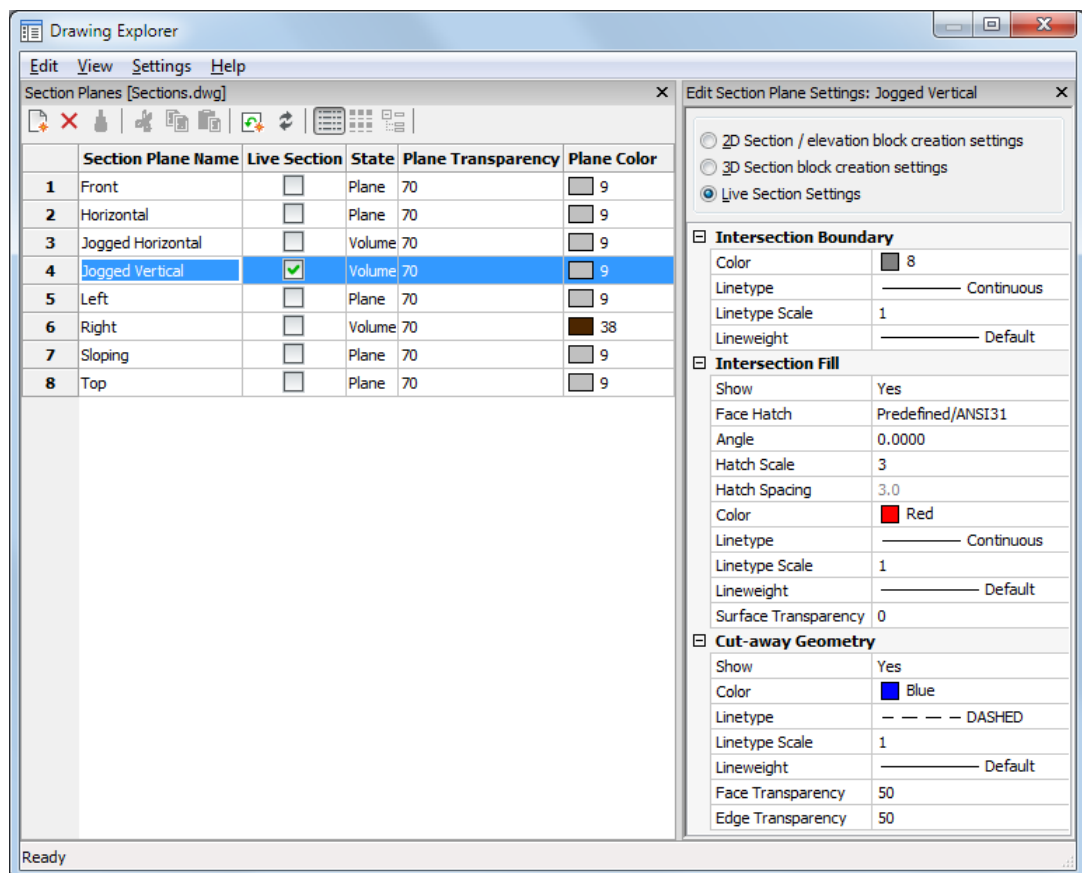
To open the Drawing Explorer - Section Planes dialog

Do one of the following:

- Choose *Drawing Explorer > Section Planes* in the *Tools* menu.
- Choose *Sections > Section Plane Settings...* in the *Model* menu.
- Type *sectionplanesettings* in the command bar, then press Enter.

To edit the properties of a section plane

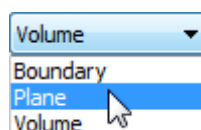
- Open the Drawing Explorer - Section Planes dialog.



- (option) Click the *Section Plane Name* field, then click again to type a new name.
- (option) Click the *Live Section* check box of a section plane to toggle the *Live Section* property of the section plane.

If necessary, click the *Regen* tool button (🔄) in the *Details* toolbar to update the display.

- (option) Click the *State* field of a section plane to choose a different state for the section plane.

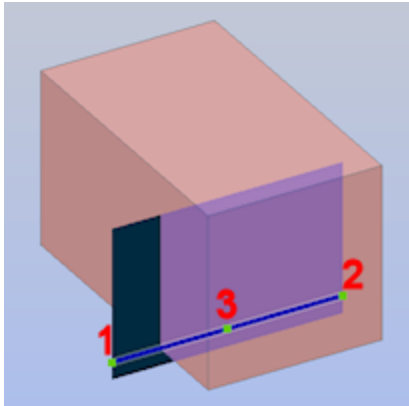


- (option) click the *Plane Transparency* field of a section plane to change the transparency of the section plane entity in the drawing.
0 = Opaque, 100 = Transparent (invisible).

6. (option) Click the *Plane Color* field, then choose a new color in the Select Color dialog.

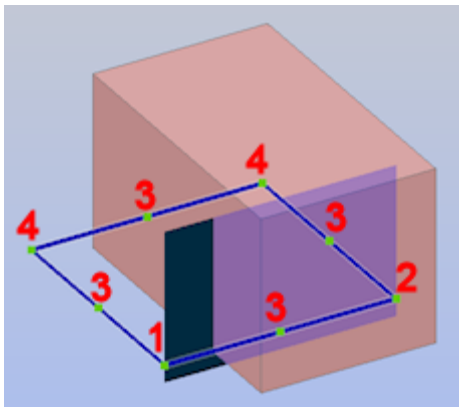
To modify the size and position of a section plane

1. Select the section plane entity in the drawing.
Depending on the current state of the section plane entity a number of grips display.
2. (option) Switch on the *Live Section* property of the section plane to see the result of the modifications dynamically.
3. (option) To grip edit a *Plane* state section plane entity:



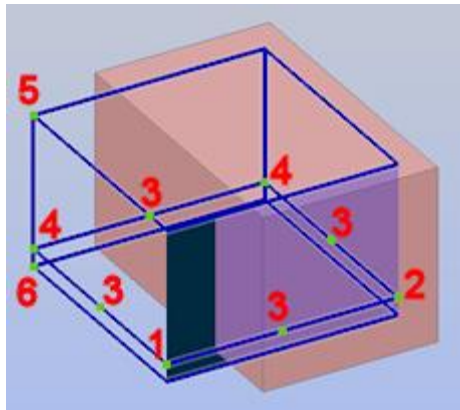
Three grips display.

- Grip 1 (= origin point) moves the position of the section plane in the XY-plane.
 - Grip 2 (= through point) redefines the through point of the section plane.
The section plane is rotated around the origin point (1).
 - Grip 3 moves the section plane parallel in a direction perpendicular to the section plane.
4. (option) To grip edit a *Boundary* state section plane entity:



Nine grips display.

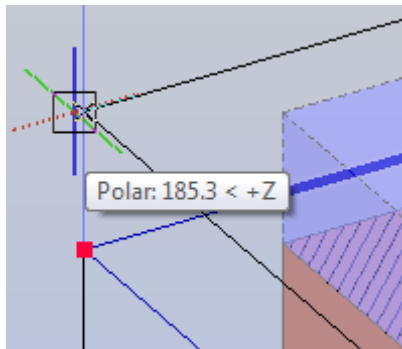
- Grip 1 (= origin point) moves the position of the section plane in the XY-plane.
 - Grip 2 (= through point) redefines the through point of the section plane.
The section plane boundary is stretched and rotated around the origin point (1).
 - Grip 3 stretches the section plane boundary parallel.
 - Grip 4 modifies the shape of the section plane boundary.
5. (option) To grip edit a *Volume* state section plane entity:



Eleven grips display.

- Grip 1 (= origin point) moves the position of the section plane in the XY-plane.
- Grip 2 (= through point) redefines the through point of the section plane.
The section plane volume is stretched and rotated around the origin point (1).
- Grip 3 stretches the section plane volume parallel.
- Grip 4 modifies the shape of the section plane volume.

Polar Tracking must be on to move grips 5 and 6.




- Grip 5 moves the top face of the section plane volume.
- Grip 6 moves the bottom face of the section plane volume.

Using Live Section

When the *Live Section* property of a section plane is on, the sectioning of a 3D solid, surface or region displays in the drawing. If the section entity is moved or modified, the sectioning is updated dynamically. The *Live Section Settings* of the section entity control the appearance of the live section.

To set the *Live Section* property of a section plane:

Method 1: Using the LIVESECTION command.

1. Do one of the following:
 - Click the *Live Section* tool button () on the *Sections* toolbar.
 - Choose *Live Section* in the *Model > Sections* menu.

The command bar reads: Select a section entity:

2. Click a section entity in the drawing.
The *Live Section* property of the selected section plane is toggled.

Method 2: Using the Properties bar.

1. Select the section plane entity in the drawing.

2. In the Properties Bar, under *Selection Entity*, click the *Live Section* field and select Yes.

Method 3: Using the *Drawing Explorer - Section Planes* dialog.

1. Select the section plane in the *Drawing Explorer - Section Planes* dialog
2. Click the check box in the *Live Section* field.

To define the live section settings

1. Open the *Drawing Explorer - Section Planes* dialog.
 2. Select the section plane.
 3. Select *Live Section Settings* in the *Edit* pane on the *Drawing Explorer - Section Planes* dialog.
- The live section settings display.

Edit Section Plane Settings: Section Plane (1)

☐ 2D Section / elevation block creation settings
☐ 3D Section block creation settings
☒ **Live Section Settings**

Intersection Boundary

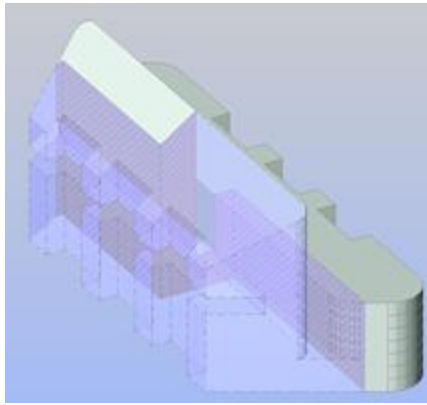
Color	8
Linetype	Continuous
Linetype Scale	1
Lineweight	Default

Intersection Fill

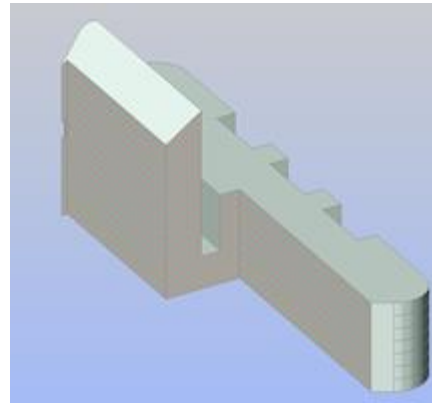
Show	Yes
Face Hatch	Predefined/ANSI31
Angle	0.0000
Hatch Scale	1
Hatch Spacing	5.0
Color	<input checked="" type="checkbox"/> ByEntity
Linetype	Continuous
Linetype Scale	1
Lineweight	Default
Surface Transparency	0

Cut-away Geometry

Show	Yes
Color	Blue
Linetype	DASHED
Linetype Scale	1
Lineweight	Default
Face Transparency	30
Edge Transparency	50



Cut-away Geometry ON



Cut-away Geometry OFF


To save a section

The SECTIONPLANETOBLOCK command allows you to:

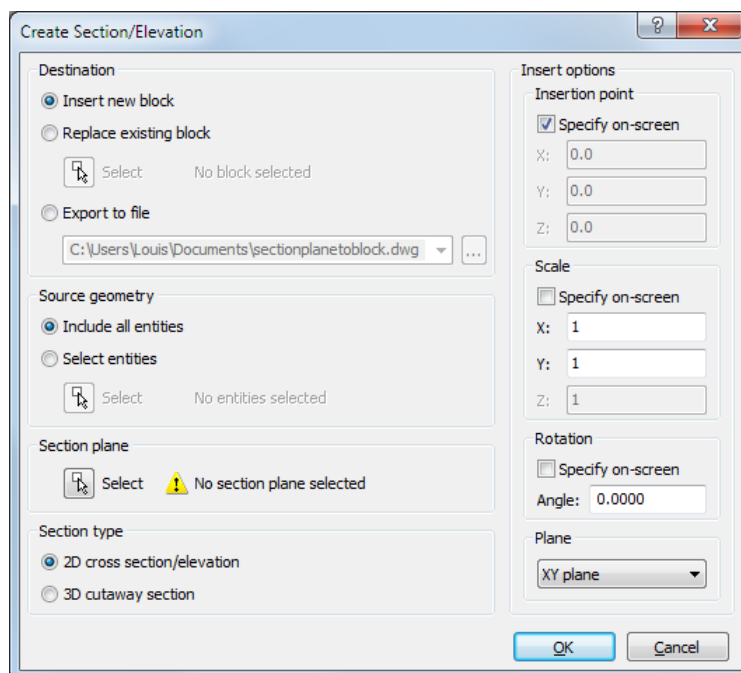
- Insert a section in the drawing as a 2D section / elevation block or a 3D cutaway block.
- Replace a section block (e.g. when the section entity has been modified).
- Save a section to a new drawing.

To launch the SectionPlaneToBlock command



Do one of the following:

- Click the *Section Plane to Block* tool button  on the *Sections* toolbar.
- Choose *Section Plane to Block* in the *Model > Sections* menu.
- Type *sectionplanetoblock* in the command bar, then press Enter



The *Create Section/Elevation* dialog box displays:






To insert a section as a 2D block

1. (option) Adjust the 2D Section / Elevation settings of the section plane entity.
2. Launch the Section Plane to Block command.
3. Set the *Destination* to *Insert new block*.
4. Under *Source Geometry*, do one of the following:
 - Choose *Include all entities* to include all 3D solids in the drawing.
 - Choose *Select entities*, then click the *Select* button () to select 3D solids.
5. Under *Section plane*, click the *Select* button () , then select a section plane entity in the drawing.
6. Set the *Section type* to *2D cross section/elevation*.
7. Adjust the *Insert* options (see Insert a block).
8. Click the *OK* button.
9. The *Create Section/Elevation* dialog closes.
The block is attached to the cursor.
Follow the instructions in the command bar to place the block in the drawing.




To insert a section as a 3D block

1. (option) Adjust the 3D cutaway section settings of the section plane entity.
2. Launch the Section Plane to Block command.
3. Set the *Destination* to *Insert new block*.
4. Under *Source Geometry*, do one of the following:
 - Choose *Include all entities* to include all 3D solids in the drawing.
 - Choose *Select entities*, then click the *Select* button () to select 3D solids.
5. Under *Section plane*, click the *Select* button () , then select a section plane entity in the drawing.
6. Set the *Section type* to *3D cutaway section*.
7. Adjust the *Insert* options (see Insert a block).
8. Click the *OK* button.
9. The *Create Section/Elevation* dialog closes.
The block is attached to the cursor.
Follow the instructions in the command bar to place the block in the drawing.

To replace an existing block

1. Launch the Section Plane to Block command.
2. Set the *Destination* to *Replace existing block*.
3. Click the *Select* button () , then select the block to be replaced in the drawing.
4. Under *Source Geometry*, do one of the following:
 - Choose *Include all entities* to include all 3D solids in the drawing.
 - Choose *Select entities*, then click the *Select* button () to select 3D solids.
5. Under *Section plane*, click the *Select* button () , then select a section plane entity in the drawing.
6. Under *Section type*, do one of the following:
 - Choose *2D cross section/elevation*, to replace the block by a 2D section / elevation block.
 - Choose *3D cutaway section*, to replace the block by a 3D cutaway section block.
7. Click the *OK* button.
The selected block is replaced.

To export a section to a file

1. Launch the Section Plane to Block command.
2. Set the *Destination* to *Export to file*.
3. Click the *Browse* button () , then do the following in the *Select export file* dialog box:
 - Select a folder.
 - Type a name in the *File name* field.
 - Click the *Save* button.
4. Under *Source Geometry*, do one of the following:
 - Choose *Include all entities* to include all 3D solids in the drawing.
 - Choose *Select entities*, then click the *Select* button () to select 3D solids.
5. Under *Section plane*, click the *Select* button () , then select a section plane entity in the drawing.
6. Under *Section type*, do one of the following:
 - Choose *2D cross section/elevation*, to replace the block by a 2D section / elevation block.
 - Choose *3D cutaway section*. to replace the block by a 3D cutaway section block.
7. Click the *OK* button.
Section is saved to a drawing.

Hatching

When you add hatching to a drawing, BricsCAD fills entities or enclosed areas with a predefined pattern or lines. First you specify the hatch pattern and other options, and then you choose which entities or enclosed areas that you want to hatch.

NOTES

- Depending on the pattern, hatchings are memory intensive and might take a considerable amount of time to draw and display. To improve performance, add hatching as one of the last steps when you create a drawing, or insert hatches on a separate layer that you can freeze as you continue to work on your drawing. Or use the *FILLMODE* system variable to switch off the display of hatches, gradient fills and filled entities.
- Hatches that are too dense, are not displayed. If such hatches exist in a drawing, a warning displays in the command bar after opening the drawing. The maximum number of dashes is controlled by the *MAXHATCH* settings variable. By default the maximum number of dashes is 100 000.

The predefined and user pattern files sit in the *Support* folder of the current user.

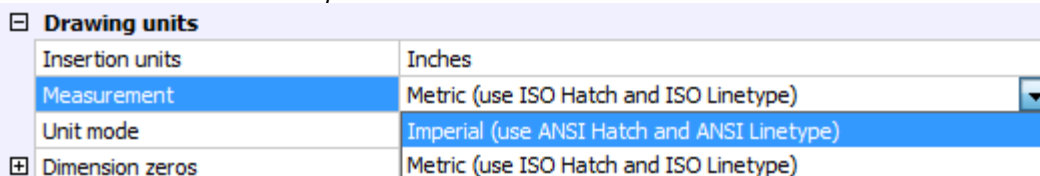
The *MEASUREMENT* setting controls which predefined hatch pattern file will be used.

- *Imperial*: uses Default.pat
- *Metric*: uses Iso.pat

Defining the MEASUREMENT setting

Do one of the following:

- In the command bar type *measurement*, then press Enter.
Type *ON*, then press Enter to set *measurement* to *Metric*.
Type *OFF*, then press Enter to set *measurement* to *Imperial*.
- In the Settings dialog go to *Drawing > Drafting > Drawing units*. Choose either *Metric* or *Imperial* in the combo box.



Defining the MAXHATCH setting

Do one of the following:

- In the command bar type *maxhatch*, then press Enter.
Type a new value and press enter.

- In the Settings dialog, go to *Drawing > Drafting > Entity Creation > Hatches*. Type a new value in the Maximum hatch dashes settings field.

[-] Hatches	
Hatch pattern angle	0
Hatch pattern boundary	Polyline
Hatch pattern doubling	<input type="checkbox"/> Hatch pattern doubling
Hatch pattern name	ANSI31
[+] Hatch pattern origin	0.0000, 0.0000
Hatch pattern scale	1.0000
Hatch pattern spacing	1.0000
Hatch pattern style	Standard
Maximum hatch dashes	100000

Creating Hatches and Gradient Fills

Commands: HATCH and GRADIENT

The *Hatch* command fills an enclosed area in your drawing with a pattern.

The *Gradient* command creates solid fills in nine different patterns and one or two colors.

The area can either be a single entity such as a circle or a closed polyline or an area enclosed by a selection of entities.

To create the hatch you can click inside the closed perimeter of a boundary or you can select entities.





Creating Hatches (Internet connection needed)



Creating Gradient Fills (Internet connection needed)

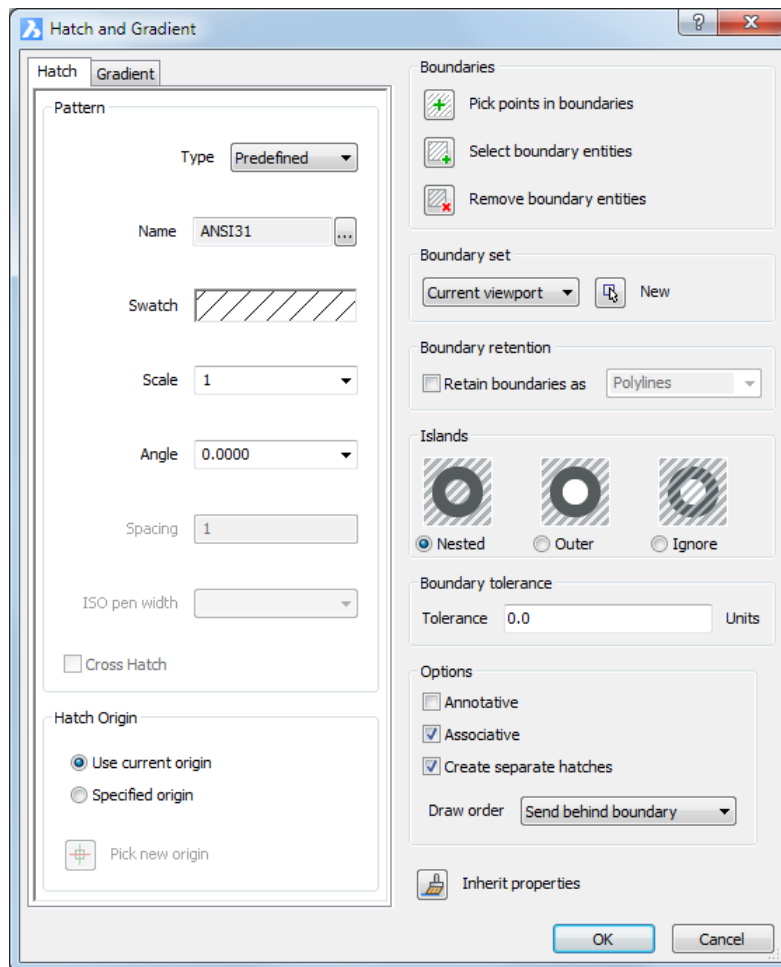
General procedure to create hatches and gradient fills

1. Do one of the following:
 - Click the *Hatch* tool button () or the *Gradient* tool button () on the *Draw* toolbar.
 - Choose *Hatch...* or *Gradient...* in the *Draw* menu.
 - Type *hatch* or *gradient* in the command bar, then press enter.

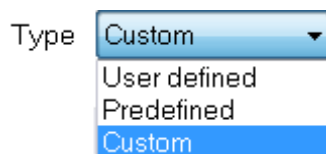
The *Hatch and Gradient* dialog displays.
2. Do one of the following:
 - Click the *Hatch* tab to create a hatch, then define the hatch properties.
 - Click the *Gradient* tab to create a gradient fill, then define the gradient fill properties.
3. Define the boundary for the hatch or gradient fill.
4. Click the *OK* button to create the hatch or gradient fill.

To define hatch properties

1. On the Hatch and Gradient dialog click the *Hatch* tab.



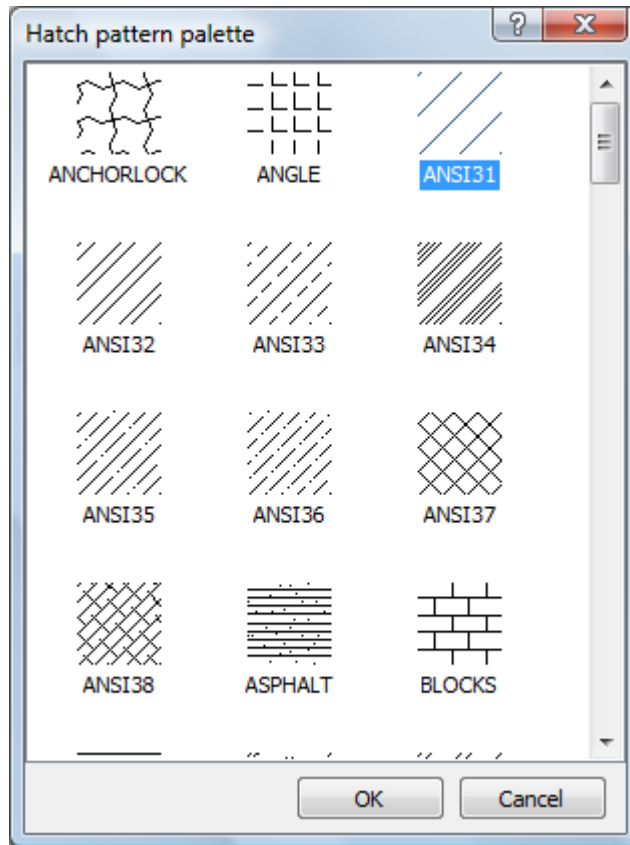
2. (option) Select the *Pattern Type*.




- **User Defined** - uses the pattern defined in this dialog box with the Angle, Spacing, and Cross Hatch options.
- **Predefined** - uses patterns defined in *iso.pat* (metric units - MEASUREMENT = ON) and *default.pat* (imperial units - MEASUREMENT = OFF).
- **Custom** - uses patterns defined in *.pat files (one pattern definition per file).

BricsCAD searches for *.pat files in the folders that are defined in the SRCHPATH (Support File Search Path) system variable.

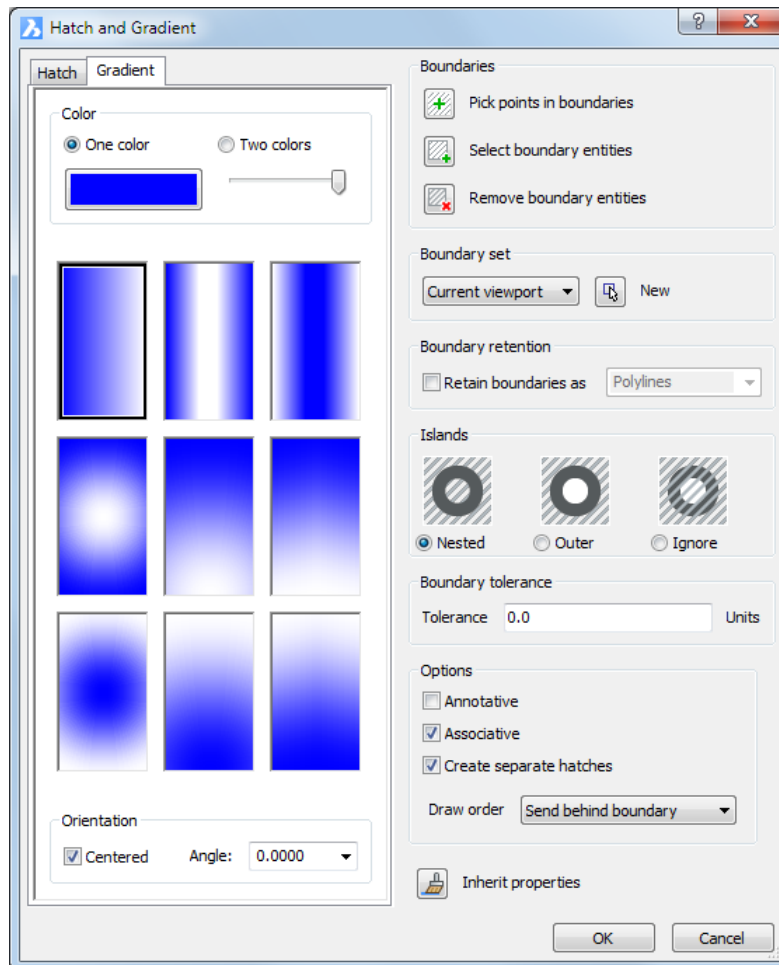
3. (option) If the *Pattern Type* is *Predefined* or *Custom* select a *Pattern Name* in the *Hatch pattern* palette, then double click the pattern or click the *OK* Button.




4. (option) If the *Pattern Type* is *Predefined* or *Custom*, do one of the following:
 - Type a scale in the *Scale* settings field.
 - Choose the scale from the drop down list.
5. (option) Do one of the following:
 - Type an angle in the *Angle* settings field.
 - Choose the angle from the drop down list.
6. (option) If the *Pattern Type* is *User defined*, specify the *Pattern Spacing*.
7. (option) If the *Pattern Type* is *User defined*, select the *Cross Hatch* option.
8. Define the Hatch Origin. The options are:
 - *Use current origin*: The origin as defined by the HPORIGIN system variable is reused.
 - *Specified origin*: Click the *Pick new origin* button (), then pick a point in the drawing.
The new origin is stored in the HPORIGIN system variable.

To define gradient fill properties

1. On the Hatch and Gradient dialog click the *Gradient* tab.



2. (option) Click the *One color* radio button to create a 'color to white' or 'color to black' gradient fill.
 - Click the colored tile to open the *Select color* dialog, to pick a different base color.
 - The color slider sets the transition from color to white or black.

Press and hold the left mouse button to move the color slider ()

Move the slider to the far right to create a 'color to white' gradient fill.
Move the slider to the far left to create a 'color to black' gradient fill.
3. (option) Click the *Two color* radio button to create 'two color' gradient fill. Click the colored tiles to open the *Select color* dialog to pick the base colors.
4. Select one of the 9 gradient patterns.
The options are: linear, cylindrical, inverted cylindrical, spherical, hemispherical, curved, inverted spherical, inverted hemispherical and inverted curved.
5. (option) *Centered*: If centered, the gradient fill is created symmetrical; if not centered, the gradient fill is built up from the left of the boundary.
6. (option) *Angle*: Specifies the angle of the gradient fill, relative to the current UCS.

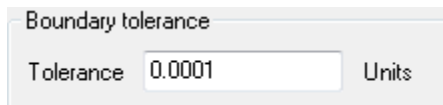
To define the boundary for a hatch or gradient fill

1. (option) Select the *Boundary retention* option.

2. (option) Select an *Island* option.

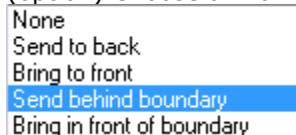


3. (option) Adjust the *Boundary tolerance*.








4. (option) Select the *Associative* option.
5. (option) Select the *Create Separate Hatches* option.
In case multiple boundaries are detected or multiple entities are selected in step 9, separate hatches are created instead of a single hatch.

6. (option) Choose a *Draw order* option.




This option is reset to *Send behind boundary* when the drawing is closed and reopened.

7. (option) Click the *Select* button () to select a *Boundary set*.
The *Hatch and Gradient* dialog temporarily closes to let you select entities.
(see also the note below)
8. (option) Click the *Specified Origin* radio button, then click the *Pick a new origin* button ().
The *Hatch and Gradient* dialog temporarily closes to let you specify a new origin.
9. Do one or more of the following:

- Click the *Pick Points* button ().
The *Hatch and Gradient* dialog closes.
The command bar reads: Select a point to define a boundary or hatch area:
 - Specify a point in each of the areas you want to hatch.
The command bar reads: Select a point to define a boundary or hatch area:
 - Right click or press Enter to stop adding areas.
The *Hatch and Gradient* dialog displays again.
- Click the *Select Boundary Entities* button ().
The *Hatch and Gradient* dialog closes.
The command bar reads: Select entities:
 - Select the entities you want to hatch.
 - Right click or press Enter to stop selecting entities.
The *Hatch and Gradient* dialog displays again.
- Click the *Remove Boundary Entities* button ().
The *Hatch and Gradient* dialog closes.
The command bar reads: Select entities:
 - Select the current boundary entities to be removed from the selection.
 - Right click or press Enter to stop selecting entities.
The *Hatch and Gradient* dialog displays again.

10. Define the Hatch Origin. The options are:

- *Use current origin*: The origin as defined by the HPORIGIN system variable is reused.
- *Specified origin*: Click the *Pick new origin* button () , then pick a point in the drawing.
The new origin is stored in the HPORIGIN system variable.

NOTE If a selection set was active when you launch the *Hatch* tool, this selection will be used as the *Boundary Set* if you click the *New* button in step 11 in the above procedure. In this case the *Hatch and Gradient* dialog closes and immediately reopens.

Editing a hatch or gradient fill

If you select a hatch or gradient fill, its properties display in the Properties bar.



To edit a hatch or gradient fill in the properties bar

1. Select the hatch or gradient fill in the drawing.

The properties of the selected entity display in the *Properties* bar:

General	
Color	<input type="checkbox"/> ByLayer
Layer	0
Linetype	ByLayer
Linetype scale	1
Plot style	ByColor
Lineweight	ByLayer
Hyperlink	
3D Visualization	
Material	ByLayer
Pattern	
Type	Predefined/ANSI31
Pattern name	ANSI31
Annotative	Yes
Annotative scale	1:1
Angle	0.0000
Scale	1
Origin point	0.0, 0.0
X	0.0
Y	0.0
Spacing	1
Double	No
Associative	Yes
Island detection	Nested
Misc	
Elevation	0.0
Area	2483.1

Hatch properties

General	
Color	<input type="checkbox"/> ByLayer
Layer	0
Linetype	———— ByLayer
Linetype scale	1
Plot style	ByColor
Lineweight	———— ByLayer
Hyperlink	
3D Visualization	
Material	ByLayer
Pattern	
Gradient name	 Linear
Annotative	Yes
Annotative scale	1:1
Origin point	0.0, 0.0
X	0.0
Y	0.0
Associative	Yes
Island detection	Nested
Color 1	 RGB:0,0,255
Color 2	<input type="checkbox"/> RGB:255,255,255
Gradient angle	0.0000
Centered	Yes
Misc	
Elevation	0.0
Area	4419.5

Gradient fill properties.

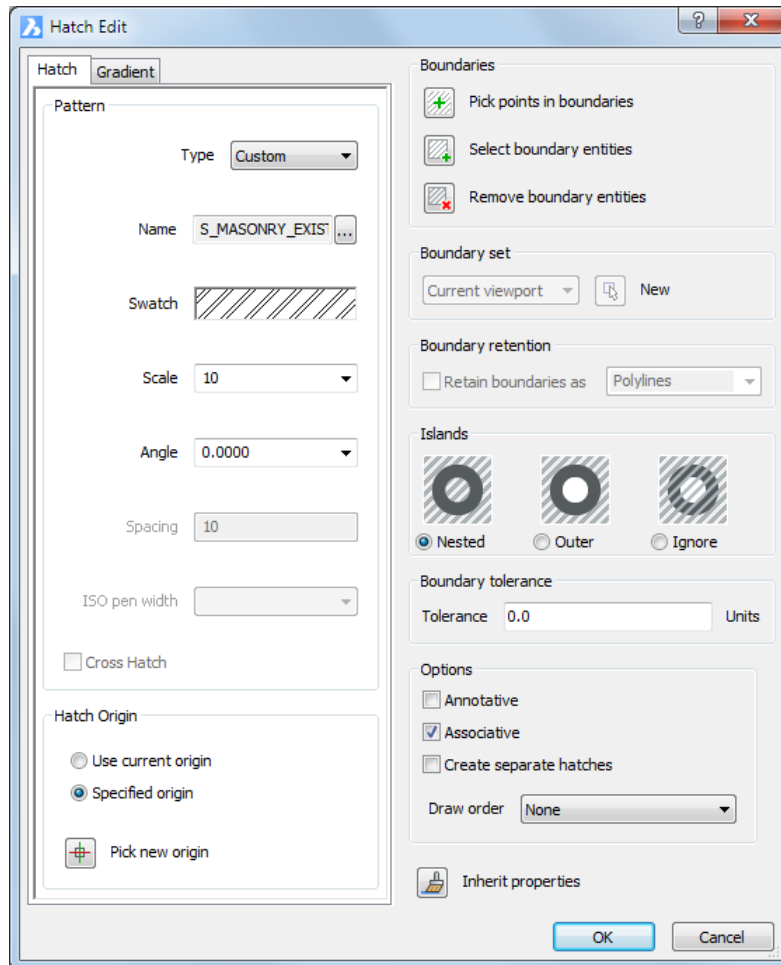
2. Select the property you want to modify in the *Properties* bar. The selected property field is activated.
3. Modify the selected property
4. Do one of the following:
 - Repeat steps 2 and 3 to modify another property.
 - Press the Esc key to stop editing the hatch.

- NOTES**
- If you select multiple hatches you can edit them simultaneously. Properties which are different display as **varies** in the BricsCAD Properties bar. If you edit such property, all edited hatches will then share this property.
 - Linetype, Linetype scale and Lineweight apply to *User Defined* pattern types only.

To edit a hatch in a dialog box

1. Do one of the following:
 - Double click a hatch or gradient fill entity.
 - Type *hatchedit* at the command prompt, then select a hatch or gradient fill entity.

The Hatch Edit dialog displays:

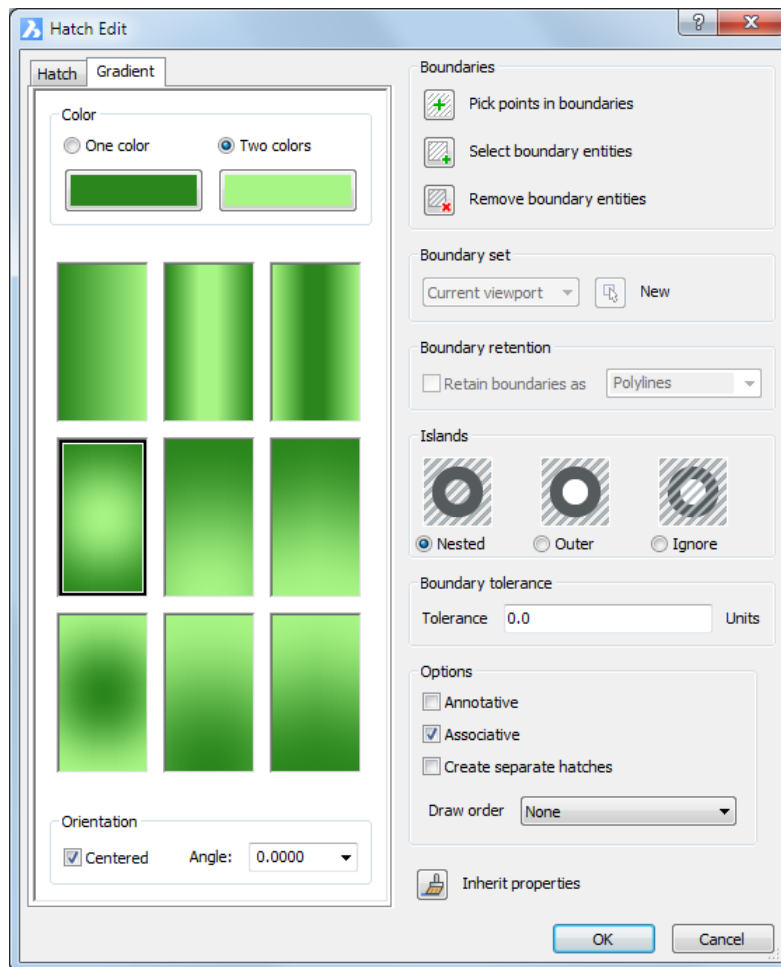


2. (option) Select the *Gradient* tab to turn the Hatch into a Gradient fill.
3. Edit the Pattern settings.
4. Edit the Boundary settings.
5. Click the *OK* button to apply the changes.

To edit a gradient fill in a dialog box

1. Do one of the following:
 - Double click a hatch or gradient fill entity.
 - Type *hatchedit* at the command prompt, then select a hatch or gradient fill entity.

The Hatch Edit dialog displays:

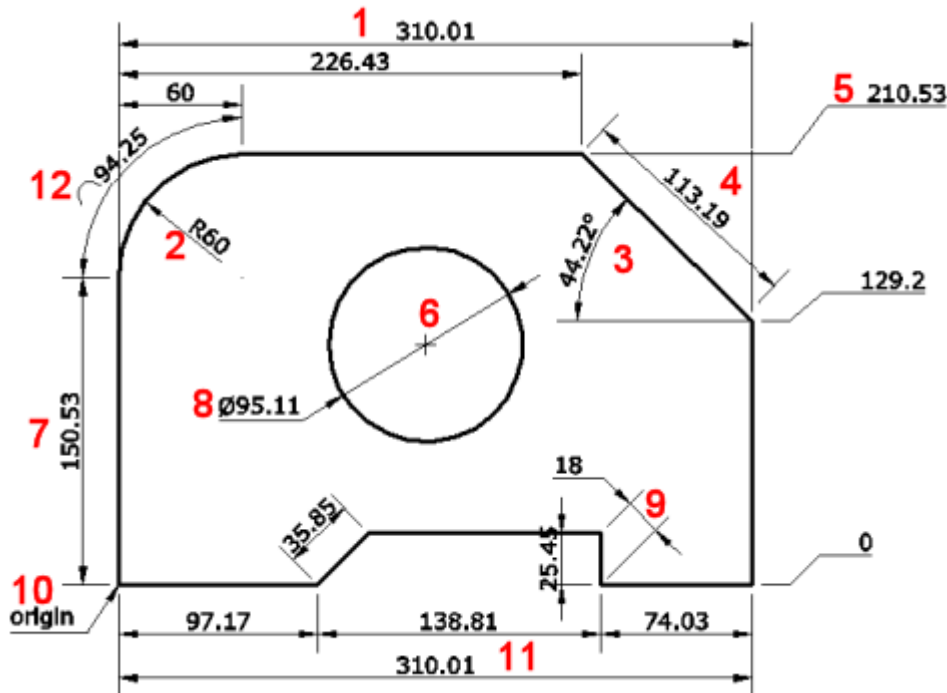


2. (option) Select the *Hatch* tab to turn the Gradient fill into a Hatch .
3. Edit the Color settings.
4. Edit the Boundary settings.
5. Click the *OK* button to apply the changes.

Dimensions

Dimensioning Concepts

In BricsCAD there are five dimension types: *linear*, *angular*, *radial*, *diametric* and *ordinate*. Dimensions are created by either selecting an entity or by identifying the origin points of the extension lines.



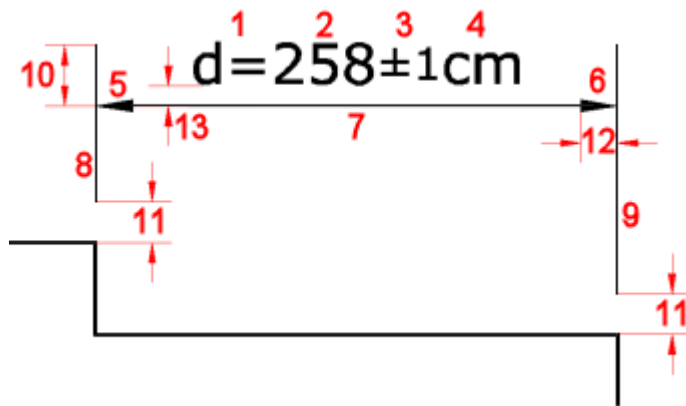
1. Stacked baseline dimension
2. Radial dimension
3. Angular dimension
4. Aligned linear dimension
5. Ordinate dimension
6. Center mark
7. Vertical linear dimension
8. Diametric dimension
9. Rotated linear dimension
10. Leader with text
11. Horizontal linear dimension
12. Arc length dimension

Dimension Style

Dimensions are created on the current layer, using the current dimension style, which controls the appearance of the dimension block. A dimension style is a collection of settings and definitions which determine the type of arrowheads, text style, color of text, lines and arrowheads, unit type, precision, etc. You can modify a dimension by editing one of the dimension settings or you can modify the dimension style to apply the changes to all dimension entities that are created with this style. A drawing contains at least one dimension style.

Dimension Block

A dimension entity is created as block, which consists of the dimension line, arrow heads, extension lines and the dimension text.



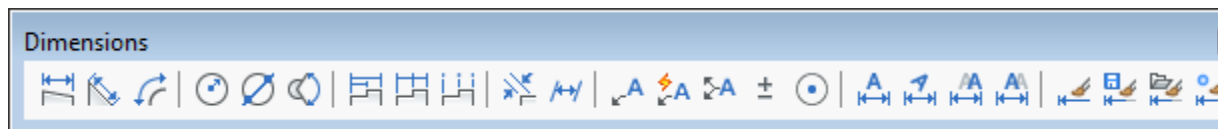
1. Dimension text prefix (d=)
2. Dimension text (258)
3. Dimension tolerance (±1)
4. Dimension text suffix (cm)
5. First arrowhead block
6. Second arrowhead block
7. Dimension line
8. First extension line
9. Second extension line
10. Extension line extend
11. Extension line offset
12. Arrowhead size
13. Vertical text offset (the vertical position of the dimension text is controlled through three settings variables: DIMGAP, DIMITAD and DIMITVP)

NOTE The Associativity (*DIMASSOC*) settings variable determines whether *associative dimensions*, *non-associative dimensions* or *exploded dimensions* are created.

Dimensioning tools overview

























All tools to create dimension entities can be found either on the *Dimensions* toolbar or in the *Dimensions* menu.

Toolbar



Menu

	Linear
	Aligned
	Arc
	Radius
	Diameter
	Angular
	Baseline
	Continue
	Ordinate
	Rotated
	Make Oblique
	Leader
	QLeader
	Multileader
	Tolerance
	Center Mark
	Edit Dimension Text
	Rotate Dimension Text
	Reposition Dimension Text
	Restore Text Position
	Apply Dimension Style
	Save Dimension Style
	Restore Dimension Style
	Dimension Variable Status

Icon	Tool name	Command	Description
	Linear	DIMLINEAR	Creates horizontal or vertical linear dimensions.
	Aligned	DIMALIGNED	Creates linear dimensions which are aligned with the dimension points.
	Rotated	DIMLINEAR	Creates rotated linear dimensions.
	Angular	DIMANGULAR	Creates angular dimensions
	Arc	DIMARC	Creates arc length dimensions
	Leader	DIMLEADER	Creates a leader with or without text.
	Quick Leader	QLEADER	Draws leaders. Lets you specify the properties of the leader through a dialog box and reuse them for a series of leaders.
	Multi-leader	MLEADER	Creates Mleaders (multi-leaders) using a multi-leader style.
	Center Lines	DIMCENTER	Draws center lines in circles and circular arcs.
	Diameter	DIMDIAMETER	Creates diametrical dimensions in circles and circular arcs.
	Radius	DIMRADIUS	Creates radial dimensions in circles and circular arcs.
	Baseline	DIMBASELINE	Creates stacked linear dimensions.
	Continue	DIMCONTINUE	Creates a chain of linear dimensions, starting from an existing linear dimension entity.
	Ordinate	DIMORDINATE	Creates ordinate dimensions.
	Tolerance	TOLERANCE	Defines the layout of geometric tolerances.
	Make Oblique	DIMEDIT + O	Makes the extension lines of the selected dimension(s) oblique.
	Edit Dimension Text	DIMEDIT+ E	Defines the dimension text of the selected dimension(s).
	Rotate Dimension Text	DIMEDIT+ R	Rotates the dimension text of the selected dimension(s).
	Reposition Dimension Text	DIMTEDIT	Lets you reposition the dimension text.
	Restore Text Position	DIMEDIT	Undoes the rotation or repositioning of the dimension text of the selected dimension(s).
	Apply Style	-DIMSTYLE + A	Applies the current dimension style to the selected dimension(s).
	Save Style	-DIMSTYLE + S	Saves the current dimension settings in a new dimension style.
	Restore Style	-DIMSTYLE	Sets the current dimension style.
	Dimension Variable Status	-DIMSTYLE + ST	Displays the status of the dimension variables in the command bar and the Prompt History window.

Dimension Styles explorer

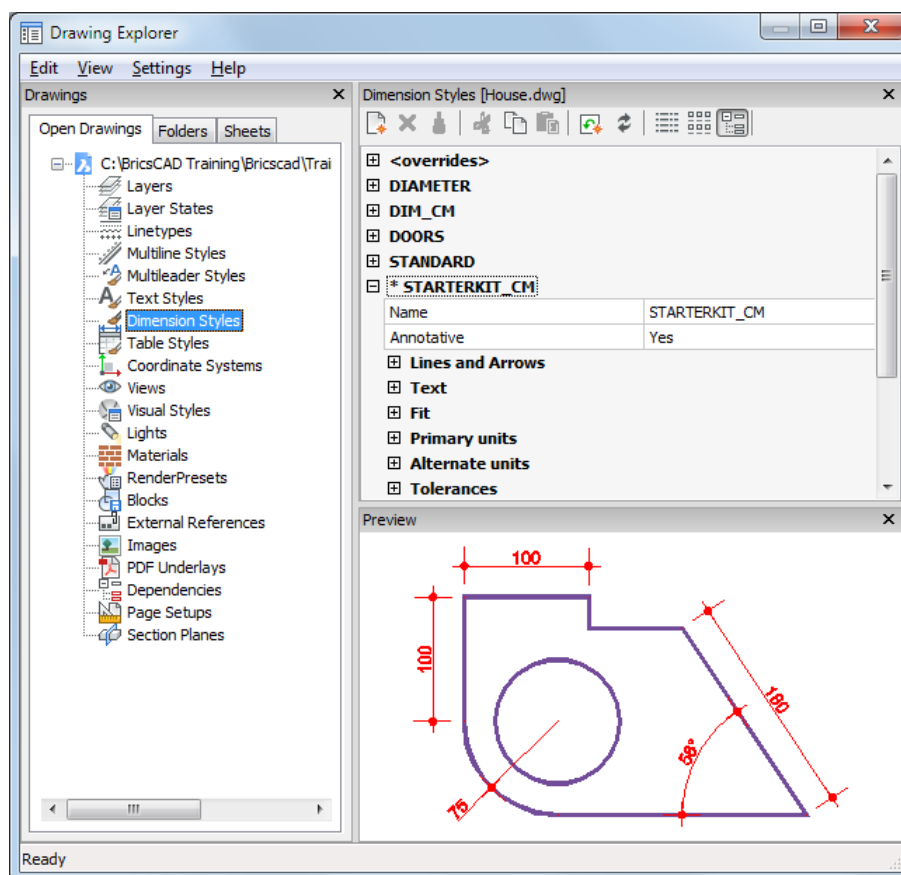
Command: DIMSTYLE

A dimension style is a collection of settings and definitions which determine the type of arrowheads, text style, color of text, lines and arrowheads, unit type, precision, etc. Dimension styles are saved in the drawing. You can create, edit and delete dimension styles. Dimension styles can be copied between drawings.

- NOTES**
- You cannot delete the current dimension style.
 - You cannot delete a dimension style that is used in the drawing.
 - Each drawing contains at least one dimension style.

Exploring dimension styles

- To open the *Dimension Styles Explorer*, do one of the following:
 - Choose *Dimension Styles ...* in the *Tools > Drawing Explorer* menu.
 - Double click the *Dimension Style* field in the Status bar.
 - Right click the *Dimension Style* field in the Status bar, then choose *Properties* in the context menu.
 - Type *dimstyle* in the command bar, then press Enter.



The current dimension style is marked with an asterisk (*).

- (option) Select a dimension style to see a preview.
- (option) Select a dimension style then right click:
 - Choose *New* to create a new dimension style.
 - Choose *Delete* to delete the selected dimension style.

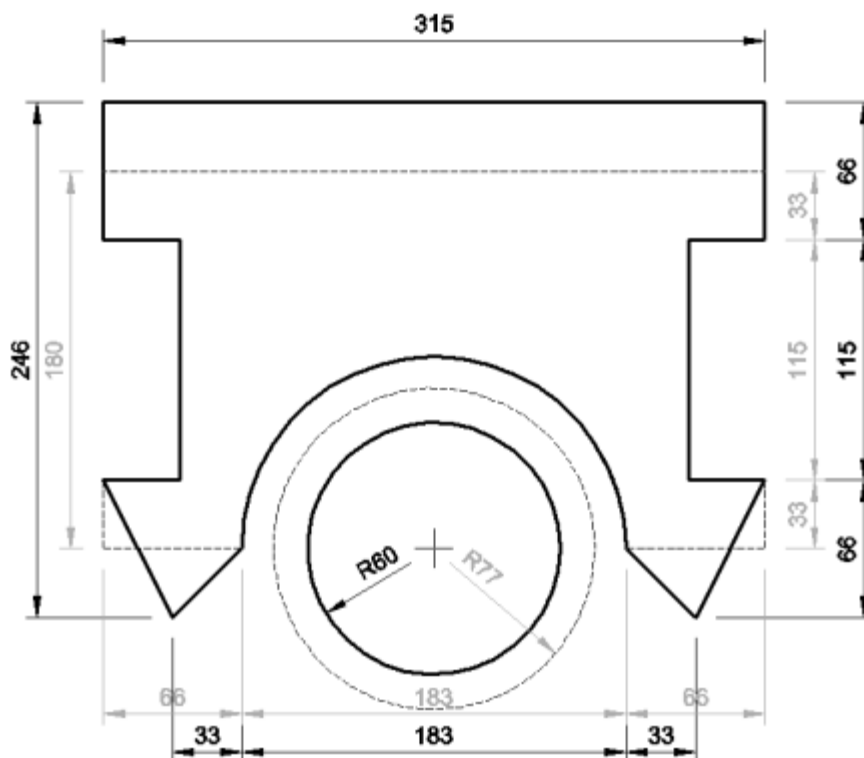
- Choose *Cut* selected dimension style in the current drawing, then paste it in another open drawing.
 - Choose *Copy*, then paste it in another open drawing.
 - Choose *Paste*, to copy a previously cut or copied dimension style in the current drawing.
 - Choose *Rename*, to rename of the selected dimension style.
 - Choose *Set current* in the context menu to make it the current dimension style.
 - Choose *Save overrides to current style*, to save all current overrides to the current style.
 - Choose *Save to new style*, to create a new dimension style as a copy of the selected dimension style.
4. (option) Modify a dimension style definition.
 5. (option) Select <overrides> to define override settings for the current dimension style.

Associative Dimensions



Creating Associative Dimensions (Internet connection needed)

Dimensions created in BricsCAD are associative: if the dimensioned entity is moved or modified, the dimensions will update automatically. The associative property of dimensions also applies to dimensions which are placed in paperspace and are associated with model space entities. If necessary, you can use the DIMREGEN command to update all associative dimensions.




The following Entity Snaps can be used to create associative dimensions:
endpoint, midpoint, center, perpendicular, quadrant, insertion, point, intersection and apparent intersection.

Dimension Settings

A dimension style consists of six settings groups

- Lines and Arrows
- Text
- Fit
- Primary Units
- Alternate Units
- Tolerances

To list the current status of the dimension variables

1. Do one of the following:
 - Click the *Dimension Variable Status* tool button  on the *Dimensions* toolbar.
 - Choose *Dimension Variable Status* in the *Dimensions* menu.

A list of all dimension variables displays in the command bar.
2. Press the *F2* function key to open the *BricsCAD Prompt History* window.
3. Press Enter to continue the listing.
4. Press the *F2* function key to close the *BricsCAD Prompt History* window.


Lines and Arrows

Name	Settings Variable	Description
Arrow 1	DIMBLK1	Defines the arrow type for the arrow of the first dimension line (Dim line 1)
Arrow 2	DIMBLK2	Defines the arrow type for the arrow of the second dimension line (Dim line 2) If DIMBLK2 equals DIMBLK1 it will be modified automatically when DIMBLK1 has changed.
Arrow size	DIMASZ	Sets the size of the arrows in drawing units.
Leader arrow	DIMLDRBLK	Defines the arrow type created by the Leader tool.
Dim line color	DIMCLRD	Sets the color of the dimension line.
Dim line LW	DIMLWD	Sets the line weight of the dimension line.
Dim line linetype	DIMLTYP	Sets the line linetype of the dimension line.
Dim line ext	DIMDLE	Defines the extension of the dimension line beyond the extension lines, when arrowheads are set to oblique or architectural ticks.
Dim baseline spacing	DIMDLI	Sets the spacing of the dimension line for stacked baseline dimensions.
Dim line 1	DIMSD1	Controls the display of the first half of the dimension line, from the first extension line to the origin point of the dimension text.
Dim line 2	DIMSD2	Controls the display of the second half of the dimension line, from the origin of the dimension text to the second extension line.
Ext line color	DIMCLRE	Sets the color of the extension lines
Ext line LW	DIMLWE	Sets the line weight of the extension lines

Ext line ext	DIMEXE	Defines the extension of the extension lines beyond the dimension line.
Ext line offset	DIMEXO	Defines the offset of the extension lines from the dimension origin points
Ext line 1	DIMSE1	Controls the display of the first extension line.
Ext line 2	DIMSE2	Controls the display of the second extension line.
Ext line type 1	DIMLTEX1	Sets the linetype for the first extension line.
Ext line type 2	DIMLTEX2	Sets the linetype for the second extension line.
Ext line fixed	DIMLFXLON	Determines whether fixed length extension lines are used.
Ext line fixed length	DIMFXL	Specifies the length of the extension lines, measured from the dimension line (= DIMEXE excl.), if DIMFXLON is on.
Center mark	DIMCEN	Defines the type of entity that will be created by the Dimcenter tool. The options are: Mark, Line, None
Center mark size	DIMCEN	DIMCEN=0: no center mark, nor center lines are drawn DIMCEN>0: a center mark is drawn, the value of DIMCEN defines the length of the center mark lines. DIMCEN<0: a center mark and center lines are drawn; the absolute value of DIMCEN defines the extension of the center lines outside the arc or circle and the gap between the center mark and the center lines.
Arc length symbol	DIMARCSYM	Controls how the arc symbol displays in an arc length dimension
Jog angle	DIMJOGANG	Specifies the angle of the oblique dimension line segment in jogged radius dimensions

NOTE Dimension lines and extension lines are created in the current line type.

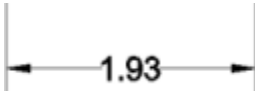
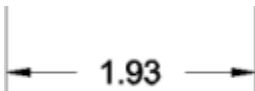
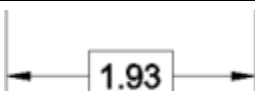
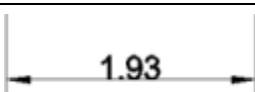
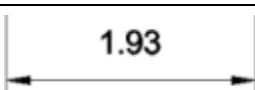
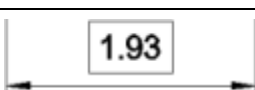
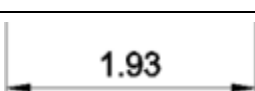
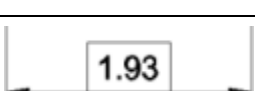
Text

Name	Settings Variable	Description
Text style	DIMTXSTY	Defines the text style of the dimension text.
Text color	DIMCLRT	Defines the color of the dimension text.
Text fill	DIMTFILL	Sets the dimension text background.  Left: <i>Background</i> = drawing window background color (<i>BKGCOLOR</i>). Right: <i>Color</i> = color defined by <i>DIMTFILLCLR</i> .
Text fill color	DIMTFILLCLR	Defines the text background color
Text height	DIMTXT	Sets the height of the dimension text in drawing units.
Draw frame around text	DIMGAP	Toggles the text frame on/off. A frame is drawn if <i>DIMGAP</i> is negative. In the Drawing Explorer > Dimensions settings dialog, the 'draw frame around text' option is enabled automatically when the value in the <i>Text offset</i> field changes from zero to a non-zero value.

Text offset	DIMGAP	Sets the offset distance around the dimension text and the distance between annotation and hook line created using the Leader command. A negative value for <i>DIMGAP</i> draws a box around the dimension or annotation text. If <i>DIMTVP</i> > 0, only negative values for <i>DIMGAP</i> (= draw text frame) make sense.
Text pos vert	DIMTAD	Defines the vertical text position with respect to the dimension line.
Text pos hort	DIMJUST	Defines the horizontal position of the text with respect to the extension lines.
Text offset vertical	DIMTVP	Sets the vertical position of dimension text above or below the dimension line when <i>DIMTAD</i> =0. The offset equals the product of <i>DIMTVP</i> and the height of the dimension text (<i>DIMTXT</i>).
Text inside align	DIMTIH	Defines whether text that is placed inside the extension lines is aligned with the dimension line or placed horizontally.
Text outside align	DIMTOH	Defines whether text that is placed outside the extension lines is aligned with the dimension line or placed horizontally.

The vertical position of the dimension text is controlled through three settings variables: DIMGAP, DIMTAD and DIMTVP.

The scheme below illustrates the compound effect of these three settings variables:

Settings	Result
DIMGAP=0 DIMTAD=0 DIMTVP=0	
DIMGAP=0.2 DIMTAD=0 DIMTVP=0	
DIMGAP=-0.1 DIMTAD=0 DIMTVP=0	
DIMGAP=0 DIMTAD=1 DIMTVP=0	
DIMGAP=0.2 DIMTAD=1 DIMTVP=0	
DIMGAP=-0.1 DIMTAD=1 DIMTVP=0	
DIMGAP=0 DIMTAD=0 DIMTVP=1	
DIMGAP=-0.1 DIMTAD=0 DIMTVP=1	

Fit

Name	Settings Variable	Description
Arrow and text fit	DIMATFIT	Determines how text and arrowheads are placed if there is not enough room to fit both, text and arrowheads between the extension lines.
Text inside	DIMTIX	Forces text to be placed inside, arrowheads outside.
Dimline inside	DIMSOXD	Suppresses the dimension lines to be drawn outside the extension lines.
Text movement	DIMTMOVE	Specifies whether the dimension line moves with the text or that text can be moved independently. This setting also specifies if a leader needs to be drawn if the text is moved away from the dimension line.
Dim scale overall	DIMSCALE	Defines a scale factor for all entities in the dimension block.
Place text manually	DIMUPT	Allows to place text manually when creating the dimension block.
Dimline forced	DIMTOFL	Forces the dimension line to be drawn, even when the text is placed outside.

Primary Units

Name	Settings Variable	Description
Dim units	DIMLUNIT	Defines the unit type for the primary units.
Dim precision	DIMDEC	Sets the precision for the primary units.
Fractional type	DIMFRAC	Determines the fractional type if <i>Dim Units</i> is <i>Fractional</i> .
Decimal separator	DIMDSEP	Sets the decimal separator character
Dim round	DIMRND	Specifies the roundoff rule for linear dimensions. E.g. if set to 0.1, all dimensions will be rounded to the nearest 0.1 unit. If set to 0, dimensions are not rounded.
Dim prefix	DIMPOST	Text string placed in front of the dimension text.
Dim suffix	DIMPOST	Text string placed after the dimension text.
Dim scale linear	DIMFLAC	Scale factor for the primary units. E.g. if your drawing units are mm and you want your dimensions to be expressed in cm, set the scale factor to 0.1.
Suppress leading zeros	DIMZIN	Suppresses the zero that is placed before the decimal separator. If on, a distance of <i>0.0124</i> will display as <i>.0124</i> .
Suppress trailing zeros	DIMZIN	Suppresses trailing zeros in decimal values. If on, a distance of <i>125,00</i> displays as <i>125</i> and <i>82.50</i> displays as <i>82.5</i> if <i>Dim Precision</i> is 2.
Suppress zero feet	DIMZIN	Suppresses zero feet values in imperial units.
Suppress zero inches	DIMZIN	Suppresses zero inches values in imperial units.
Dim angle units	DIMAUNIT	Sets the unit for angular dimensions
Dim angle precision	DIMADEC	Sets the precision for angular dimensions
Suppress angle	DIMAZIN	Suppresses leading zeros in angular dimensions

leading zeros		
Suppress angle trailing zeros	DIMAZIN	Suppresses trailing zeros in angular dimensions

Alternate Units

Name	Settings Variable	Description
Alt enabled	DIMALT	Adds alternate units to the dimension text.
Alt units	DIMALTU	Sets the unit type for the alternate units.
Alt precision	DIMALTD	Sets the precision for the alternate units.
Alt scale factor	DIMALTF	Sets the scale factor for the alternate units. E.g. if the primary units are imperial and the alternate units are decimal (mm), then the alternate units scale factor must be set to 25.4.
Alt round	DIMALTRND	Roundoff rule for the alternate units in linear dimensions. E.g. if set to 0.1, alternate units will be rounded to the nearest 0.1 unit. If set to 0, alternate units are not rounded.
Alt prefix	DIMAPOST	Text string placed in front of the alternate dimension text.
Alt suffix	DIMAPOST	Text string placed after the alternate dimension text.
Alt suppress leading zeros	DIMALTZ	Suppresses the zero that is placed before the decimal separator in alternate units. If on, a distance of <i>0.0124</i> will display as <i>.0124</i> .
Alt suppress trailing zeros	DIMALTZ	Suppresses trailing zeros in decimal values in alternate units. If on, a distance of <i>125,00</i> displays as <i>125</i> and <i>82.50</i> displays as <i>82.5</i> if <i>Dim Precision</i> is 2.
Alt suppress zero feet	DIMALTZ	Suppresses zero feet values in imperial units.
Alt suppress zero inches	DIMALTZ	Suppresses zero inch values in imperial units.

Tolerances

Name	Settings Variable	Description
Tolerance display	DIMTOL	Adds tolerances to dimension text. If DIMTOL is On, DIMLIM is switched Off.
Limits display	DIMLIM	Sets the default text to dimension limits. If DIMLIM is On, DIMTOL is switched Off.
Tolerance precision	DIMTDEC	Sets the precision for dimension tolerances.
Tolerance limit lower	DIMTM	Sets the lower limit for dimension tolerances.
Tolerance limit upper	DIMTP	Sets the upper limit for dimension tolerances.
Tolerance text height	DIMTFAC	Sets the text height for the tolerance settings with respect to the dimension text height. E.g. if set to 0.5, the tolerance text height is half the height of the dimension text.
Tolerance pos vert	DIMTOLJ	Determines the vertical position for dimension tolerances with respect to the dimension text.

Tolerance suppress leading zeros	DIMTZIN	Suppresses leading zeros.
Tolerance suppress trailing zeros	DIMTZIN	Suppresses trailing zeros.
Alt suppress zero feet	DIMTZIN	Suppresses zero feet values in imperial units.
Alt suppress zero inches	DIMTZIN	Suppresses zero inch values in imperial units.
Alt tolerance precision	DIMALTTD	Sets the precision for dimension tolerances in alternate units.
Alt tolerance suppress leading zeros	DIMALTTZ	Suppresses leading zeros.
Alt tolerance suppress trailing zeros	DIMALTTZ	Suppresses trailing zeros.
Alt tolerance suppress zero feet	DIMALTTZ	Suppresses zero feet values in imperial units.
Alt tolerance suppress zero inches	DIMALTTZ	Suppresses zero inch values in imperial units.

Dimension Styles explorer

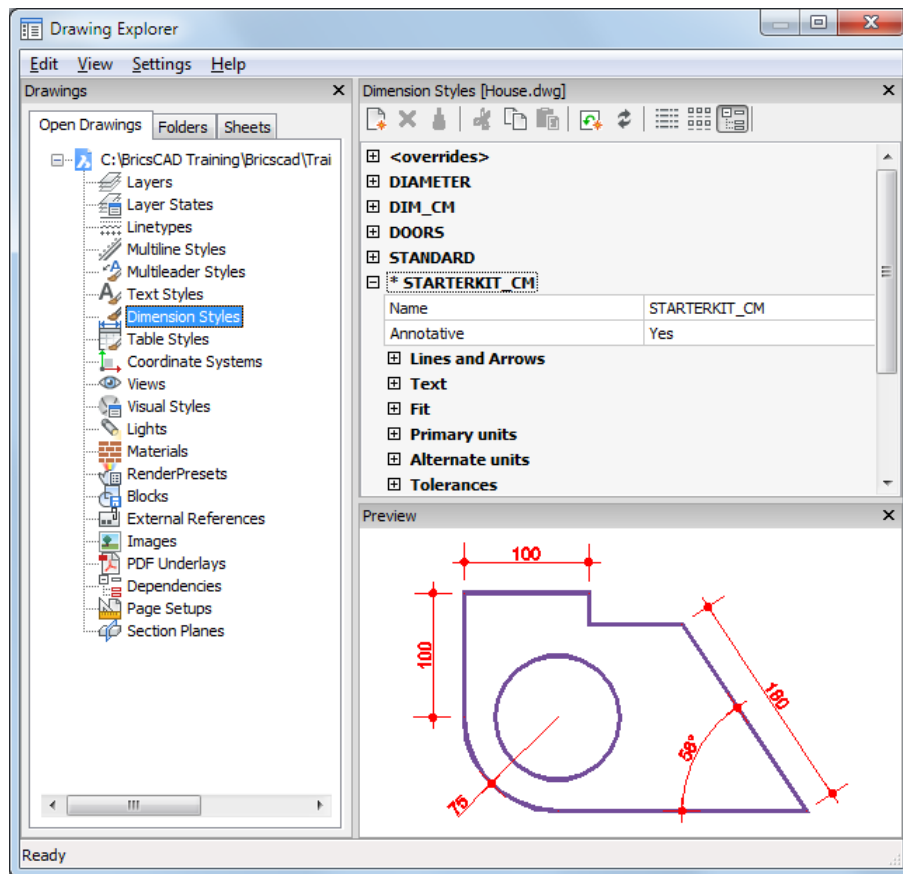
Command: DIMSTYLE

A dimension style is a collection of settings and definitions which determine the type of arrowheads, text style, color of text, lines and arrowheads, unit type, precision, etc. Dimension styles are saved in the drawing. You can create, edit and delete dimension styles. Dimension styles can be copied between drawings.

NOTES You cannot delete the current dimension style.
 You cannot delete a dimension style that is used in the drawing.
 Each drawing contains at least one dimension style.

Exploring dimension styles

- To open the *Dimension Styles Explorer*, do one of the following:
 - Choose *Dimension Styles ...* in the *Tools > Drawing Explorer* menu.
 - Double click the *Dimension Style* field in the Status bar.
 - Right click the *Dimension Style* field in the Status bar, then choose *Properties* in the context menu.
 - Type *dimstyle* in the command bar, then press Enter.



The current dimension style is marked with an asterisk (*).

2. (option) Select a dimension style to see a preview.
3. (option) Select a dimension style then right click:
 - Choose *New* to create a new dimension style.
 - Choose *Delete* to delete the selected dimension style.
 - Choose *Cut* selected dimension style in the current drawing, then paste it in another open drawing.
 - Choose *Copy*, then paste it in another open drawing.
 - Choose *Paste*, to copy a previously cut or copied dimension style in the current drawing.
 - Choose *Rename*, to rename of the selected dimension style.
 - Choose *Set current* in the context menu to make it the current dimension style.
 - Choose *Save overrides to current style*, to save all current overrides to the current style.
 - Choose *Save to new style*, to create a new dimension style as a copy of the selected dimension style.
4. (option) Modify a dimension style definition.
5. (option) Select <overrides> to define override settings for the current dimension style.

Creating linear dimensions

Commands: DIMLINEAR, DIMALIGNED, DIMBASELINE, DIMCONTINUE, and DIMARC

Linear dimensions annotate linear distances or lengths and can be oriented either horizontally, vertically or aligned parallel to an existing entity or to the selected origin points.


Starting from a linear dimension, you can add a *baseline* dimension or a *continued* dimension.

A baseline dimension inserts an additional dimension from a common first extension line origin of a previous linear dimension.

A continued dimension continues a linear dimension from the second extension line of a previous linear dimension.

To create a horizontal or vertical linear dimension

1. Do one of the following:

- Click the *Linear* tool button () on the *Dimensions* toolbar.
- Choose *Linear* in the *Dimensions* menu.
- Type *dimlinear* in the command bar, then press Enter.

The command bar reads: ENTER to select entity/<Origin of first extension line>:

2. Do one of the following:

- Press Enter, then select the line or linear segment of a polyline you want to dimension.
- Specify the origin of the first extension line, then specify the origin of the second extension line.


The dimension line displays dynamically.

Move the cross hairs vertically to create a horizontal dimension, or horizontally to create a vertical dimension.

3. Position the dimension line.

To create an aligned linear dimension

1. Do one of the following:

- Click the *Aligned* tool button () on the *Dimensions* toolbar.
- Choose *Aligned* in the *Dimensions* menu.
- Type *dimaligned* in the command bar, then press Enter.

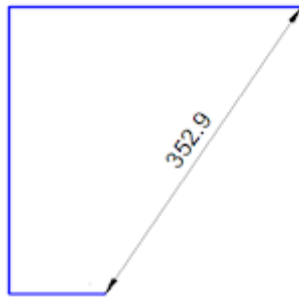
The command bar reads: ENTER to select entity/<Origin of first extension line>:

2. Do one of the following:

- Press Enter, then select the line or linear segment of a polyline you want to dimension.
- Specify the origin of the first extension line, then specify the origin of the second extension line.

3. Do one of the following:

- Click a point to position the dimension line.
- Press 0 (zero) to place the dimension line at the dimension points, thus omitting the extension lines.



To create a rotated linear dimension

1. Do one of the following:

- Click the *Rotated* tool button () on the *Dimensions* toolbar.
- Choose *Rotated* in the *Dimensions* menu.

The command bar reads: ENTER to select entity/<Origin of first extension line>:

2. Do one of the following:
 - Press Enter, then select the line or linear segment of a polyline you want to dimension.
 - Specify the origin of the first extension line, then specify the origin of the second extension line.

The command bar reads: Angle of dimension line <0>:


3. Do one of the following:
 - Type the rotation angle in the command bar, then press Enter.
 - Enter the rotation angle by specifying two points.

The dimension line displays dynamically.

4. Position the dimension line.
You can place the dimension line either at the specified rotation angle or perpendicular to this direction.

To create baseline dimensions

1. Do one of the following:

- Click the *Baseline* tool button () on the *Dimensions* toolbar.
- Choose *Baseline* in the *Dimensions* menu.
- Type *dimbaseline* in the command bar, then press Enter.

The command bar reads: Baseline: ENTER to select starting dimension/<Origin of next extension line>:

A baseline dimension displays dynamically starting from the most recently added linear or angular dimension (if any).

2. (option) Press enter, then select the starting dimension (see note below).
3. Specify the origin of the next extension line.


The command bar reads: Baseline: ENTER to select starting dimension/<Origin of next extension line>:

4. Do one of the following:
 - Repeat steps 2 and 3.
 - Press the Esc key to stop creating baseline dimensions.

- NOTES** When selecting the starting dimension make sure to click the first extension line or the first half of the dimension line in order to use the first origin point of the starting dimension as the origin point for the baseline dimension.
- If you select an angular dimension, stacked angular dimensions are created.
- The distance between the stacked dimension lines is controlled by the *Dim baseline spacing* setting in the dimension style.

To create continued dimensions

1. Do one of the following:

- Click the *Continue* tool button () on the *Dimensions* toolbar.
- Choose *Continue* in the *Dimensions* menu.
- Type *dimcontinue* in the command bar, then press Enter.

The command bar reads: Continue: ENTER to select starting dimension/<Origin of next extension line>:

A continued dimension displays dynamically starting from the most recently added linear or angular dimension (if any).

2. (option) Press enter, then select the starting dimension (see note below).
3. Specify the origin of the next extension line.

The command bar reads: Continue: ENTER to select starting dimension/<Origin of next extension line>:


4. Do one of the following:

- Repeat steps 2 and 3.
- Press the Esc key to stop creating continued dimensions.

- NOTE** When selecting the starting dimension make sure to click the second extension line or the second half of the dimension line in order to use the second origin point of the starting dimension as the origin point for the continued dimension. Otherwise the new dimension line will partly overlap the starting dimension.
- If you select an angular dimension, continued angular dimensions are created.

To create an arc length dimension

1. Do one of the following:

- Click the *Arc* tool button () on the *Dimensions* toolbar.
- Choose *Arc* in the *Dimensions* menu.
- Type *dimarc* in the command bar, then press Enter.

The command bar reads: Select arc or polyline arc segment:

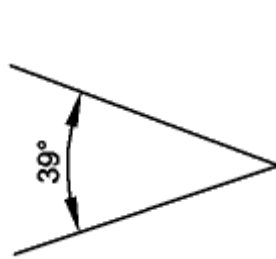
2. Select the arc or polyline arc segment.
The arc length dimension displays dynamically.
3. Position the dimension line.

Creating angular dimensions

Command: DIMANGULAR

Angular dimensions annotate the angle measured between two lines, two linear segments of a polyline or the angle of a circular arc. You can also dimension an angle by selecting an angle vertex and two endpoints.

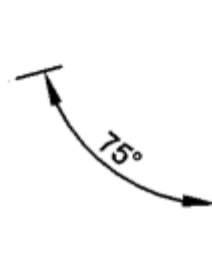
Starting from an existing angular dimension, you can add a baseline dimension or a continued dimension. An angular baseline dimension inserts an additional dimension from a common first extension line origin. An angular continued dimension continues an angular dimension from the second extension line of a previous angular dimension.



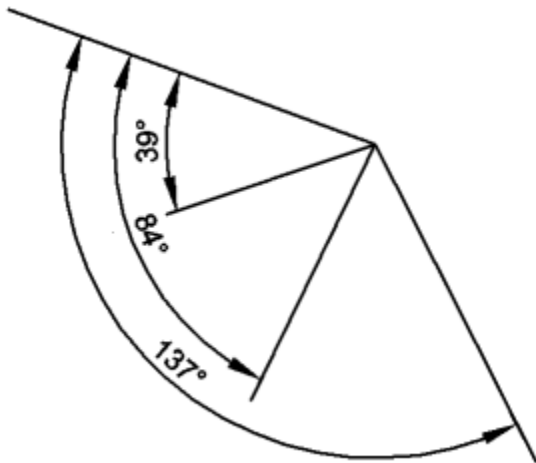
*angular dimension
measured between two
lines*



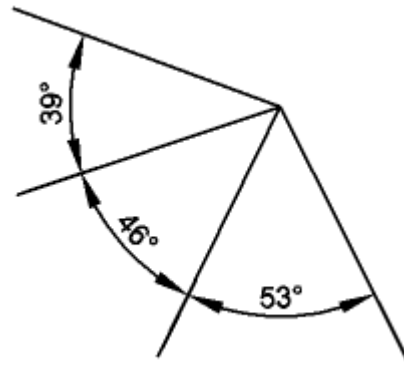
*angular
dimension of
an arc*



*angular dimension
defined by a
vertex and two
endpoints*




angular baseline dimensions



angular continued dimensions

To dimension an angle between two lines


1. Do one of the following

- Click the *Angular* tool button () on the *Dimensions* toolbar.
- Choose *Angular* in the *Dimensions* menu.
- Type *dimangular* in the command bar, then press Enter.

The command bar reads: Press Enter to specify angle/<select line, arc, or circle>:


2. Select the first line or line segment.
The command bar reads: Other line for angular dimension:
3. Select the second line or line segment.
The angular dimension displays dynamically.
The command bar reads: Angle/Text/<Location of dimension arc>:
4. Do one of the following:
 - Position the dimension arc.
 - Type *A* and press Enter to define the angle of the dimension text, then position the dimension arc.
 - Type *T* and press Enter to modify the dimension text, then position the dimension arc.

To dimension an angle encompassed by an arc

1. Do one of the following
 - Click the *Angular* tool button () on the *Dimensions* toolbar.
 - Choose *Angular* in the *Dimensions* menu.
 - Type *dimangular* in the command bar, then press Enter.

The command bar reads: Press Enter to specify angle/<select line, arc, or circle>:
2. Select the arc.
The angular dimension displays dynamically.
The command bar reads: Angle/Text/<Location of dimension arc>:
3. Do one of the following:
 - Position the dimension arc.
 - Type *A* and press Enter to define the angle of the dimension text, then position the dimension arc.
 - Type *T* and press Enter to modify the dimension text, then position the dimension arc.

To create an angular dimension defined by a vertex and two endpoints

1. Do one of the following
 - Click the *Angular* tool button () on the *Dimensions* toolbar.
 - Choose *Angular* in the *Dimensions* menu.
 - Type *dimangular* in the command bar, then press Enter.

The command bar reads: Press Enter to specify angle/<select line, arc, or circle>:
2. Right click or press Enter.
The command bar reads: Vertex of angle.
3. Specify the vertex of the angle.
The command bar reads: first side of angle.
4. Specify the endpoint of the first side of the angle.
The command bar reads: Other side of angle.
5. Specify the endpoint of the second side of the angle.
The angular dimension displays dynamically.
The command bar reads: Angle/Text/<Location of dimension arc>:


6. Do one of the following:
 - Position the dimension arc.
 - Type *A* and press Enter to define the angle of the dimension text, then position the dimension arc.
 - Type *T* and press Enter to modify the dimension text, then position the dimension arc.

Creating diametrical and radial dimensions

Commands: DIMDIAMETER and DIMRADIUS


Diametrical and radial dimensions annotate the radii and diameters of circles and circular arcs. The DIMCENTER command creates center marks in the center point of circles and circular arcs.

To create a diametrical dimension

1. Do one of the following
 - Click the *Diameter* tool button () on *Circles* toolbar or the *Circles* flyout of the *Dimensions* toolbar.
 - Choose *Diameter* in the *Dimensions* menu.
 - Type *dimdiameter* in the command bar, then press Enter.


The command bar reads: Select arc or circle
2. Select the arc or circle you want to annotate.
The diametrical dimension displays dynamically.
3. Position the dimension line.

To create a radial dimension

1. Do one of the following
 - Click the *Radius* tool button () on *Circles* toolbar or the *Circles* flyout of the *Dimensions* toolbar.
 - Choose *Radius* in the *Dimensions* menu.
 - Type *dimradius* in the command bar, then press Enter.

The command bar reads: Select arc or circle
2. Select the arc or circle you want to annotate.
The radial dimension displays dynamically.
3. Position the dimension line.

Creating center marks or center lines

1. Do one of the following
 - Click the *Center* tool button () on *Circles* toolbar or the *Circles* flyout of the *Dimensions* toolbar.
 - Choose *Center* in the *Dimensions* menu.
 - Type *dimcenter* in the command bar, then press Enter.

The command bar reads: Select arc or circle to dimension:
2. Select the arc or circle.
The center marks are created.

NOTE The *Center mark* setting of the dimension style controls whether center marks or center lines are created. The *Center mark size* settings of the dimension style controls the size of the center mark. The center mark settings are saved to the *DIMCEN* system variable.


Creating ordinate dimensions

Command: DIMORDINATE

An ordinate dimension annotates the perpendicular distance from an origin or base point (the origin of the current user coordinate system [UCS]). Ordinate dimensions consist of an x- or y-coordinate and a leader. An x-ordinate dimension measures distances along the x-axis, while a y-ordinate dimension measures distances along the y-axis.

As you select ordinate points, the program automatically determines whether the point is an x- or y-ordinate based on which direction you drag the second point. You can also specify whether the ordinate represents an x- or y-ordinate. Ordinate dimension text is always aligned with the ordinate leader lines, regardless of the text orientation specified by the current dimension style.

To create an ordinate dimension

1. Do one of the following:
 - Click the *Ordinate* tool button () on the *Dimensions* toolbar.
 - Choose *Ordinate* in the *Dimensions* menu.
 - Type *dimordinate* in the command bar, then press Enter.

The command bar reads: Select point for ordinate dimension:
2. Specify the origin point for the ordinate dimension.
The ordinate dimension displays dynamically.
3. Position the ordinate dimension.

NOTE Adjust the origin point of the current user coordinate system (UCS) before placing ordinate dimensions if necessary.

Creating leaders and annotations


Commands: DIMLEADER, LEADER, QLEADER and MLEADER

A leader consists of an arrowhead, a line, polyline or a spline and an annotation.

The general procedure to place an annotation is:

- Specify the position of the arrowhead.
- Create the leader line segments.
- Type the annotation text.

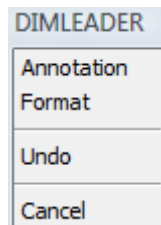
To create a leader and an annotation

1. Do one of the following:
 - Click the *Leader* tool button () on the *Dimensions* toolbar.

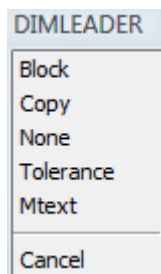
- Choose *Leader* in the *Dimensions* menu.
- Type *leader* or *dimleader* in the command bar, then press Enter.

The command bar reads: Start of leader

2. Specify the position of the arrowhead.
The command bar reads: Next point.
3. Specify the endpoint of the first leader segment.
The command bar reads: To point: Format/Undo/<Annotation>:
A prompt menu displays:




4. Add more segments to the leader, then press Enter or right click to stop adding segments.
The command bar reads: First line of annotation text/<options>:
5. Do one of the following:
 - Type the first line of the annotation text, then press Enter.
The command bar reads: Next line of annotation text:
Do one of the following:
 - Press Enter or right click to create the leader and annotation.
 - Keep adding lines to the annotation text, then press Enter or right click to create the leader and annotation.
 - Press Enter or right click to see the annotation text options.
The command bar reads: Dimension text options:
Block/Copy/None/Tolerance/<Mtext>:
A prompt menu displays:



- Press the Esc key to abort the *Leader* tool.

To create a leader only

1. Do one of the following:
 - Click the *Leader* tool button () on the *Dimensions* toolbar.
 - Choose *Leader* in the *Dimensions* menu.
 - Type *dimleader* in the command bar, then press Enter.

The command bar reads: Start of leader
2. Specify the position of the arrowhead.
The command bar reads: Next point.
3. Specify the endpoint of the first leader segment.
The command bar reads: To point: Format/Undo/<Annotation>:

4. Add more segments to the leader, then press Enter or right click to stop adding segments.
The command bar reads: First line of annotation text/<options>:
5. Press Enter, then choose *None* in the prompt menu or type *N*.

Editing dimensions

Commands:

DIMEDIT: Changes the wording and angle of dimension text; changes the angle of extension lines.

DIMTEDIT: Changes the position of dimension text.

Dimension entities can be edited in the following ways:

- use grips to edit dimensions
- edit the dimension text
- rotate dimension text
- reposition the dimension text
- restore the text position
- make the extension lines oblique

Using grips to edit a linear dimension

1. Select the dimension entity.

5 grips display:

- 2 origin point grips (1)
- 2 dimension line grips (2)
- 1 dimension text grip (3)



2. (option) Drag the origin point grips to adjust the dimension origin points.
The dimension text is adjusted automatically.
3. (option) Drag dimension line grips to move the dimension line.
4. (option) Drag the text grip to move the dimension text.

NOTES Depending on the *Text movement* setting, the dimension line will move along with the text or the text will move independently. A leader will be drawn if the *Text movement* setting is set to *Move text, add leader*.

If two adjacent dimensions (continued dimensions) are selected both, you can move their coinciding grips simultaneously.

Editing the dimension text

Dimension text can be edited:

- in the MText editor:
 - using the DDEDIT command
 - double clicking the dimension entity
- in the *Text override* field in the Properties Bar.
- using the *Edit Dimension Text* tool.

NOTES The following tags can be used to format the dimension text:

- type \L (backslash upper case L) to start underlined text, \l (backslash lower case L) to stop underlined text
 - type <> to add the default dimension text
 - type [] to add the alternate units, if alternate units is currently switched off in the dimension style.
 - type %%d to add the degree symbol (°)
 - type %%c to add the diameter symbol (Ø)
 - type %%p to add the plus/minus symbol (±)
- e.g. if you type %%c\L<> cm in the *Text override* field, dimension text displays as: Ø 24,43 cm


To edit the dimension text in the MText editor

1. Double click the dimension entity.
The *Text Formatting* toolbar displays.
2. Edit the dimension text.
3. Do one of the following:
 - Click the *OK* button on the *Text Formatting* toolbar.
 - Click outside the dimension text.

To edit the dimension text in the Properties Bar

1. Select the dimension entity.
2. In the Properties Bar, select the *Text override* field.
3. Type the new dimension text in the *Text override* field and press Enter.
4. Press the Esc key to stop editing the dimension entity.


To edit the dimension text using the Edit Dimension Text tool

1. Do one of the following:
 - Click the *Edit Dimension Text* tool () in the *Dimensions* toolbar.
 - Choose *Edit Dimension Text* in the *Dimensions* menu.
 - Type *dimedit* in the command bar, press Enter, then type *new* and press Enter.

The command bar reads: New dimension text <>:


2. Type the new dimension text and press Enter.
The command bar reads: Select dimensions to replace with new text:
3. Select the dimensions and press Enter.

To rotate the dimension text

1. Do one of the following:
 - Click the *Rotate Dimension Text* tool () in the *Dimensions* toolbar.
 - Choose *Rotate Dimension Text* in the *Dimensions* menu.
 - Type *dimedit* in the command bar, press Enter, then type *R* and press Enter.


The command bar reads: Angle of dimension text:
2. Type the rotation angle, then press Enter.
The command bar reads: Select entities to rotate text:
3. Select the dimensions and press enter.

To reposition the dimension text

1. Do one of the following:
 - Click the *Reposition Dimension Text* tool () in the *Dimensions* toolbar.
 - Choose *Reposition Dimension Text* in the *Dimensions* menu.
 - Type *dimtedit* in the command bar, then press Enter.


The command bar reads: Select dimensions to reposition text:
2. Select the dimension.
The dimension text moves dynamically.
3. Click to reposition the dimension text.

To restore the text position

1. Do one of the following:
 - Click the *Restore Text Position* tool () in the *Dimensions* toolbar.
 - Choose *Restore Text Position* in the *Dimensions* menu.
 - Type *dimedit* in the command bar, then press Enter.

The command bar reads: Select dimensions to restore text to default position:
2. Select the dimensions and press enter.

To make the extension lines oblique

1. Do one of the following:
 - Click the *Make Oblique* tool () in the *Dimensions* toolbar.
 - Choose *Make Oblique* in the *Dimensions* menu.
 - Type *dimedit* in the command bar, press Enter, then type *O* and press Enter.

The command bar reads: Select dimensions to make oblique:
2. Select the dimensions and press enter.
The command bar reads: Enter obliquing angle:
3. Type the obliquing angle, then press Enter.

NOTE To restore non-rotated extension lines, execute the above procedure and type 0 (zero) when asked to enter the obliquing angle.

Adding Geometric Tolerances

Command: TOLERANCE

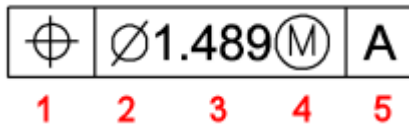
Geometric tolerances specify the maximum variation that is allowed in form or position from true geometry. The geometric tolerance is, in essence, the tolerance zone within which a surface or axis of hole or cylinder can lie which results in resulting feature being acceptable for proper function and interchangeability.

BricsCAD draws geometric tolerances using a feature control frame, which is a rectangle divided into compartments.

Each feature control frame consists of at least two compartments. The first compartment contains a geometric tolerance symbol that indicates the geometric characteristic to which the tolerance is applied, such as location, orientation, or form. For example, a form tolerance may indicate the flatness or roundness of a surface.

The second compartment contains the tolerance value. When appropriate, the tolerance value is preceded by a diameter symbol and followed by a material condition symbol. The material conditions apply to features that can vary in size.

The tolerance value can then be followed by primary, secondary, and tertiary datum reference letters, along with the material conditions of each datum. Datum reference letters are generally used as reference tolerances to one of up to three perpendicular planes from which a measurement is made, although datum reference letters can also indicate an exact point or axis.



1. Geometric characteristic symbol
2. Diameter symbol
3. Tolerance value
4. Material condition symbol
5. Datum reference

When two tolerances apply to the same geometry, you can also add a composite tolerance consisting of a primary tolerance value followed by a secondary tolerance value. To make a tolerance even more specific, it can also contain a projected tolerance consisting of a height value followed by a projected tolerance symbol. For example, you can use a projected tolerance to indicate the perpendicularity of an embedded part.

To create a geometrical tolerance frame

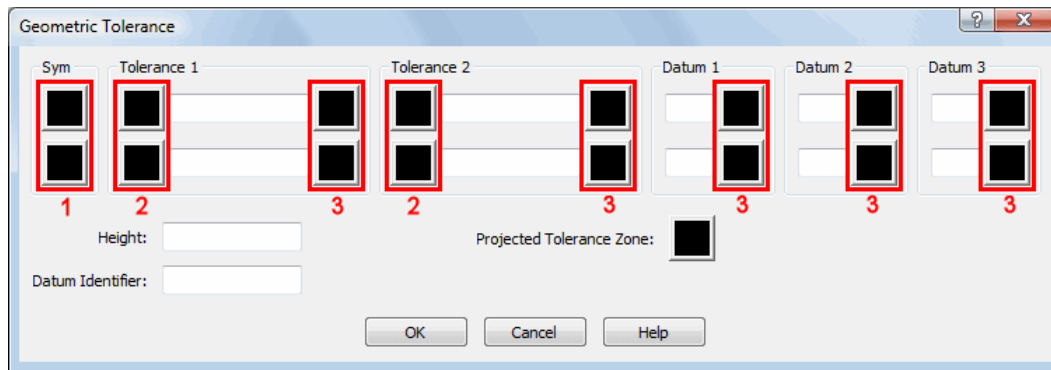
1. Do one of the following

- Click the *Tolerance* tool button (\pm) in the *Dimensions* toolbar.
- Choose *Tolerance* in the *Dimensions* menu.
- Type *tolerance* in the command bar, then press Enter.

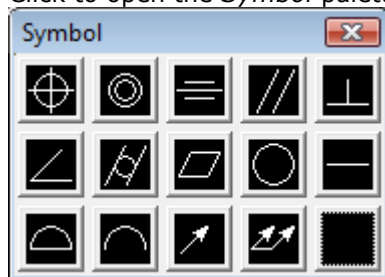
The Geometric Tolerance dialog opens.

2. Compose the geometrical tolerance frame in the *Geometric Tolerance* dialog.
3. Click the *OK* button.
The *Geometric Tolerance* dialog closes.
4. Click to insert the geometric tolerance frame in the drawing.

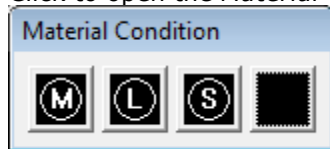
Geometric Tolerance dialog



1. *Geometric Tolerance Symbol* buttons.
Click to open the *Symbol* palette.



2. *Diameter Symbol* option buttons.
Click to add the diameter symbol.
3. *Material Condition* buttons.
Click to open the *Material Condition* palette.



Working with Texts

BricsCAD provides a variety of tools that deal with adding and editing text. You can work with single lines of text in your drawings, as well add multiple paragraphs .

The TEXT command creates single line text, while the MTEXT command creates text entities which consist of multiple lines.

Text Styles

A text style determines the appearance of the text characters in your drawing. It defines text properties such as the font, height and obliquing angle.

New text is created using the current text style by default, but you can choose a different text style in the insert text procedures or assign a different text style when editing the text.

Text Font

Two font types can be used: *SHX* fonts and *TTF* fonts.

SHX font (compiled shape font) files sit in the *Fonts* folder of the BricsCAD installation folder (e.g. C:\Program Files\Bricsys\BricsCAD\Fonts).

TTF (True Type Font) files sit in the *Fonts* folder of Windows (e.g. C:\Windows\Fonts).

Text Variables

System variables can be set using the Settings dialog.

Some system variables which apply to text entities are:

Variable Name	Variable Title	Description
FONALT	Alternate font	Defines the font which will be used if a text font cannot be found.
HIDETEXT	Hide text on hide	Specifies whether the HIDE command processes text entities.
MIRRTEXT	Mirror text	Determines whether text is mirrored by the MIRROR command or not.
MTEXTED	Multiline text editor	Sets the primary and secondary text editors to use for multiline text entities.
MTEXTFIXED	Multiline text fixed	Controls whether BricsCAD zooms, rotates and/or pans the view to fit the Mtext being edited.
QTEXTMODE	Quick text mode	Controls how text entities are displayed. If ON, displays text as a box.
TEXTANGLE	Text angle	Stores the angle of the last added text entity.
TEXTFILL	Text fill	Determines whether True Type fonts display either filled or outlined. Filled Text Outlined Text
TEXTQLTY	Text quality	Defines the smoothness of True Type fonts for plotting and rendering.

TEXTSIZE	Text size	Sets the default height for new text entities. TEXTSIZE has no effect if the current text style has a fixed height
TEXTSTYLE	text style	Stores the name of the current text style.
TSPACEFAC	Text space factor	Specifies the line spacing distance of multiline text measured as factor of the text height. Values between 0.25 and 4 are accepted.
TSPACETYPE	Text space type	Specifies the type of line spacing used for multiline text. At least: adjusts line spacing based on the tallest character(s) in a line. Exactly: used the specified line spacing, regardless of individual character sizes.
TSTACKALIGN	Text stack align	Determines the vertical alignment of stacked text: bottom, center or top.
TSTACKSIZE	Text stack size	Specifies the percentage a which the stacked text fraction height is relative to the current height of the selected text. Values between 25 and 125 are accepted.

Text styles

Command: STYLE

When you add text to a drawing, it uses the current text style, which determines the font, size, angle, orientation and other properties of the new text entity.

Every drawing has a default text style, named *Standard*, which initially uses the *txt.shx* font. You cannot delete or rename the *Standard* style, but you can modify it.

If you change the *font* or *orientation* of an existing style, all existing text entities created using that style are automatically updated to reflect the change. Changing any other property has no effect on existing text.

You can create and use an unlimited number of additional text styles.


To open the Text Style Explorer dialog

Do one of the following:

- In the *Tools > Drawing Explorer* menu choose *Text Styles ...*
- Double click the *Style* field in the Status Bar.
- Type *style* in the command bar and press Enter.

The *Drawing Explorer - Text Styles* dialog opens.

To create a text style




1. Open the Text Style Explorer dialog.
2. Do one of the following:
 - Click the *New* tool button () in the *Details* section of the *Drawing Explorer* dialog.
 - Select an existing style, then right click and choose *New* in the context menu.

A *NewStyle* text style is created, using default properties.
3. Type a name in the *Text Style Name* field, replacing the *NewStyle* default name.
4. Define the text style properties.
5. Close the *Text Style Explorer* dialog.

To edit a text style

1. Open the text style explorer dialog.
2. Select the text style.
3. Define the text style properties.
4. Close the *Text Style Explorer* dialog.


To make a text style current

1. Open the text style explorer dialog.
2. Do one of the following:
 - When in *Detail view* () , click the blank tile in front of the text style name. A blue filled circle indicates the text style is set current.
 - When in *Detail view* () or *Icon view* () , right click then choose *Set current* in the context menu.

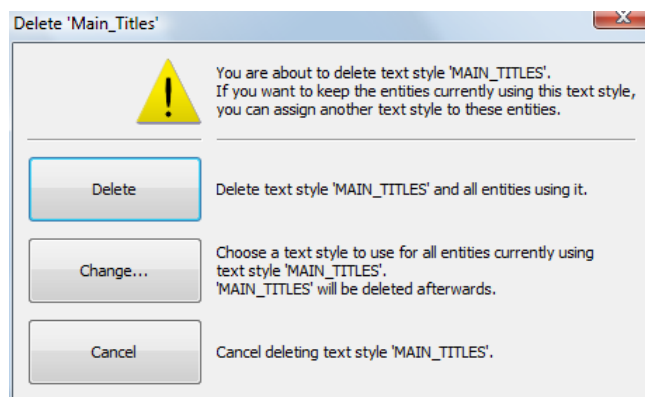
NOTE Alternatively you can right click the *Text style* field in the Status Bar, then choose the new current style. The current style is marked in the style list.

When you use the *Style* option in the create text procedures, the selected style will become the current text style.

To delete a text style

1. Open the text style explorer dialog.
2. Select the text style.
3. Do one of the following:
 - Click the *Delete* tool button () in the *Details* section of the *Drawing Explorer* dialog.
 - Right click, then choose *Delete* in the context menu.

If the text style is not used in the drawing or in a dimension style, the text style is deleted,
else an alert box displays:



4. Do one of the following:
 - Click the *Delete* button to delete the text style and all entities using it.
 - Click the *Change...* button to delete the text style and assign a different text style to the entities, which currently use the selected style.
 - Click the *Cancel* button to cancel the delete procedure.


NOTE It is not possible to delete the current text style.

Creating text

Commands: TEXT and MTEXT

The *Text* command creates single line text, while the *Mtext* command creates text entities which consist of multiple lines.

To create single line text

1. Do one of the following:
 - Click the *Text* tool button () on the *Draw* toolbar.
 - Choose *Text* in the *Text* submenu in the *Draw* menu.
 - Type *text* in the command bar, then press Enter.

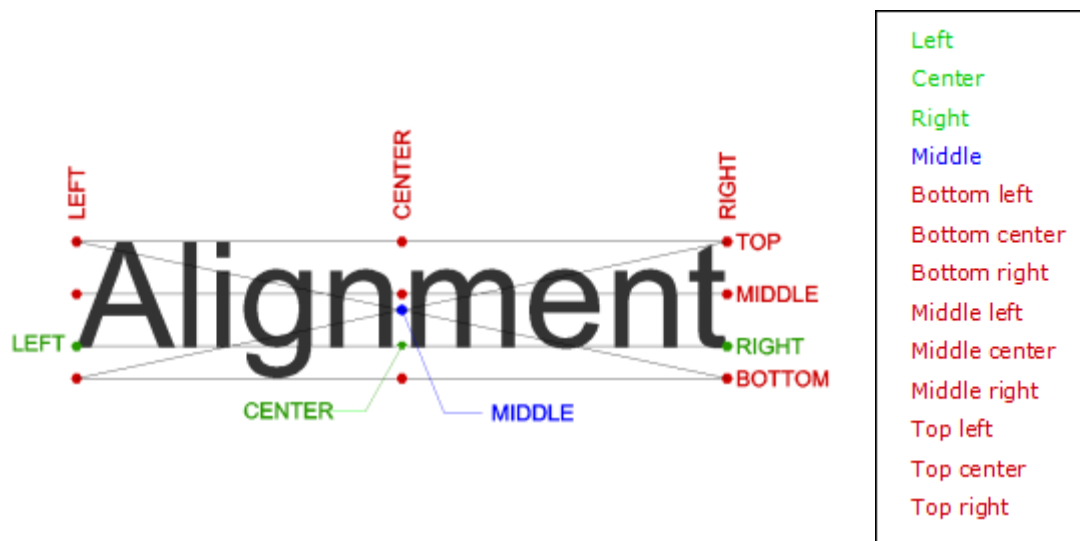
The command bar reads: Text: Style/Align/Fit/Center/Middle/Right/Justify/<Start point>:
2. Specify the insertion point of the text.
The command bar reads: Height of text <current height>:
3. Do one of the following:
 - Press Enter or right click to accept the current text height.
 - Type the new text height in the command bar, then press Enter.
 - Define the new text height graphically by specifying two points in the drawing.

The command bar reads: Rotation angle of text <current angle>
4. Do one of the following:
 - Press Enter or right click to accept the current angle.
 - Type the new angle in the command bar, then press Enter.
 - Define the now angle graphically by specifying two points in the drawing.


The command bar reads: Text:
5. Type the text in the command bar, then press Enter.
The text is placed in the drawing.
The command bar reads: Text:
6. Do one of the following:
 - Repeat step 5 to create another text entity.
The new text is placed below the previous one.
 - Press Enter or right click to conclude the *Text* command.

NOTE If you have already created text, the most recent text entity highlights when you repeat the *Text* command. If you want to place the new text right below the previously created text, press Enter in step 2 of the above procedure.

Single line text justification

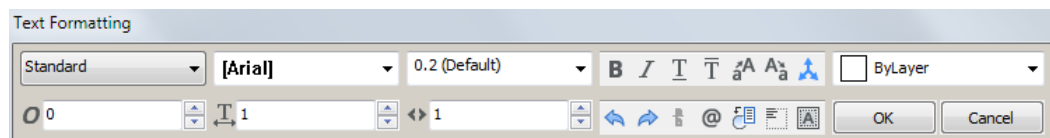


To create multiline text

1. Do one of the following:
 - Click the *Multiline Text* tool button () on the *Draw* toolbar.
 - Choose *Multiline Text* in the *Text* submenu in the *Draw* menu.
 - Type *mtext* in the command bar, then press Enter.

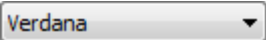





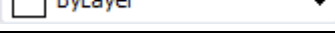
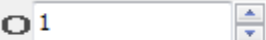







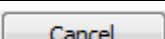
The command bar reads: First corner for block of text:
2. Specify the first corner (insertion point of the multiline text).
The command bar reads:
Justification/Rotation/Style/Height/Direction/Width/<Opposite corner for block of text>:
3. Do one of the following:
 - Specify the opposite corner of the text block to define the maximum width of the multiline text.
 - Type @ in the command bar, then press Enter for non-wrapped (zero width) text.

The *Text formatting* toolbar opens right above the text block.



4. (option) Define the text properties.
5. Type your text.
6. Click the *OK* button to stop editing the text.
The *Text formatting* toolbar closes.

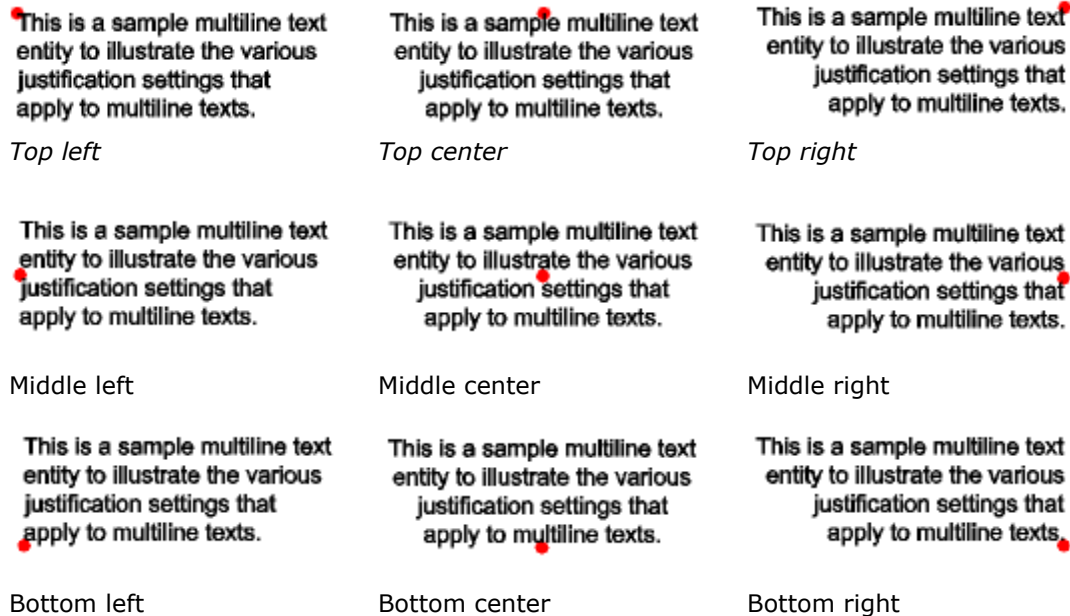
Text formatting toolbar tools and settings

Button or setting	Name	Description
	Text Style	Sets the text style
	Text Font	Sets the text font
	Text Height	Sets the text height
	Bold	Creates bold text
	Italic	Creates italic text
	Underscore	Creates underscored text
	Overscore	Creates overscored text
	Uppercase	Converts selected text to uppercase
	Lowercase	Converts selected text to lowercase
	Annotative	Toggles the Annotative property
	Layer	Sets the text layer
	Oblique Angle	Sets the oblique angle
	Width Factor	Sets the width factor
	Character Spacing	Sets the character spacing
	Undo	Undoes recent actions
	Redo	Redoes previously undone actions
	Stacked Fraction	Creates stacked fractions
	Special Character	Inserts special characters
	Field	Inserts a variable field
	Justify	Sets the text justification
	Background Mask	Adds a colored background
	OK	Closes the <i>Text formatting</i> toolbar and applies the changes
	Cancel	Closes the <i>Text formatting</i> toolbar and leaves the Mtext entity unchanged

NOTE If the line length exceeds the *Width* of the multiline text, the text will be wrapped. Set the *Width* to zero for non-wrapped multiline text.

Multiline text justification

(The red dot indicates the insertion point of the multiline text entity)



Including special text characters and symbols

1. You can use control codes and unicode strings to include special characters in text entities.
2. To include control codes, as you type text, type two percent symbols (%%) followed by the special control code or character. A single percent sign is treated as a normal text character.

Special text characters


Control code	Unicode String	Result
%%d	\U+00B0	Draws a degree symbol (°)
%%p	\U+00B1	Draws a plus-or-minus symbol (±)
%%c	\U+2205	Draws a diameter symbol (Ø)

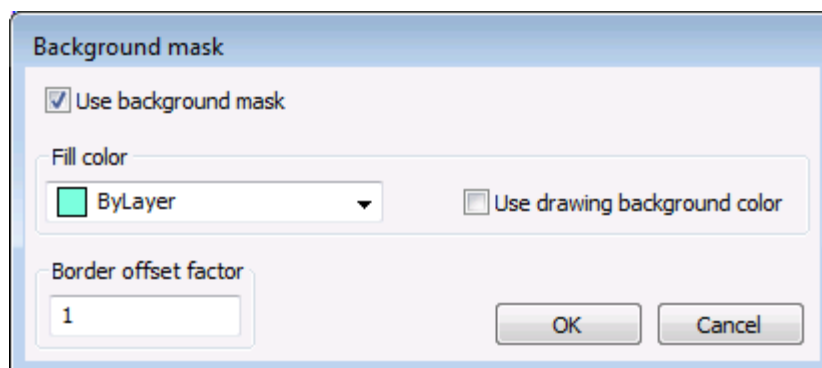
Symbols and Unicode strings

Name	Symbol (*)	Unicode String
Almost equal	≈	3. \U+2248
Angle	∠	4. \U+2220
Center line	⌀	\U+2104
Delta	Δ	\U+0394
Electrical phase	Φ	\U+0278
Identity	≡	\U+2261
Not equal	≠	\U+2260
Omega	Ω	\U+03A9
Squared	²	\U+00B2
Cubed	³	\U+00B3

5. (*) If supported by the text font.

To add a background mask

- When creating or editing a multiline text entity, click the *Background Mask* tool button () on the *Text Formatting* toolbar. The *Background Mask* dialog displays:



- Check the *Use background mask* option.
- Do one of the following:
 - Choose a color in the *Fill color* drop-down list.
 - Choose *Select color...* in the *Fill color* drop-down list and choose an Index color or a True color in the *Select color* dialog.
 - Check the *Use drawing background color* option.
- Define the *Border offset factor*, which defines the margin around the text for the background. The value is based on the text height and must be in the range 1-5. A factor of 1.0 exactly fits the multiline text entity.
- Click *OK* to create the background mask.

Editing text

Command: DDEDIT

You can edit and modify a text entity as you would any other drawing entity. That is, you can delete, move, rotate, and scale text.

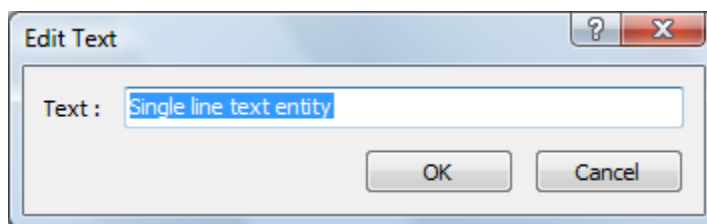
You can change the text color or layer in the Properties Bar.

To edit a multiline text, you can use the built-in text editor or choose an external program, such as Wordpad.

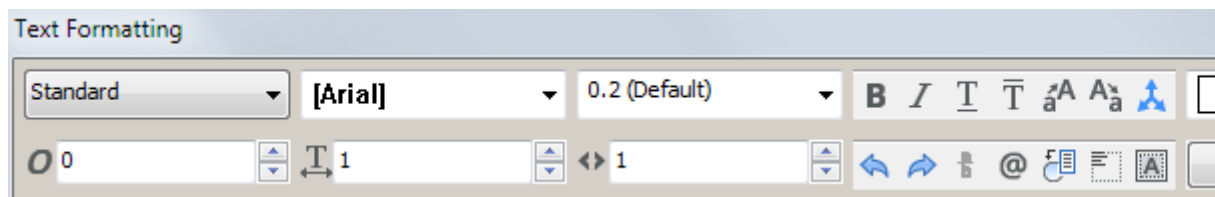
The MTEXTFIXED system variable controls whether the view is zoomed, rotated and/or panned to fit the edited Mtext.

To edit a text entity

1. Do one of the following:
 - Type *ddedit* in the command bar.
The command bar reads: Select entity to modify:
 - Double click the text entity.
Then go to step 3.
2. Do one of the following:
 - Click a single line text entity.
The *Edit Text* window opens.




- Click a multiline text entity
The *Text formatting* toolbar opens.

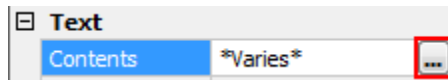


3. Edit the text.
4. Press the *OK* button on the *Edit Text* window or *Text Formatting* toolbar to apply the changes.
5. Do one of the following:
 - Select another text entity to edit.
 - Right click or press Enter to stop editing text entities.

To edit a selection of Mtext entities

1. Select the Mtext entities.
2. In the Properties bar select *Contents*.

3. Press the *Browse* button () on the right hand side of the *Contents* field



4. The *Text formatting* toolbar opens for the first Mtext entity in the selection.
5. Edit the Mtext,
6. Press the *OK* button or click in the drawing.
The next Mtext entity in the selection can be edited now.
7. Repeat steps 5 and 6 until the last Mtext is edited.

To select a different multiline text editor

Edit the *MTEXTED* system variable under *Drawing/Drafting/Texts/Multiline Texts* in the Settings dialog.

or

1. Type *mtexted* in the command bar, then press Enter.
The command bar reads: New value for MTEXTED, or . for none/<current value>:
2. Type the file name and full path of the alternate text editor in the command bar, then press Enter.
e.g. C:\Program Files\Windows NT\Accessories\Wordpad.EXE

NOTE To restore the built-in text editor, type a . (period) in step 2 of the above procedure, then press Enter.

Exploding text

Command: TXTEXP

The *TXTEXP* command explodes Text and Mtext entities into 2D polylines.

To explode text

1. Type *txtexp* in the command bar, then press Enter.
The command bar reads: Select text to be exploded:
2. Select a single text entity or multiple text entities, then press Enter.
The selected text(s) are exploded.

NOTE The solid fill of certain TTF fonts is removed when exploding such text, leaving the text outline only.

Finding and replacing text

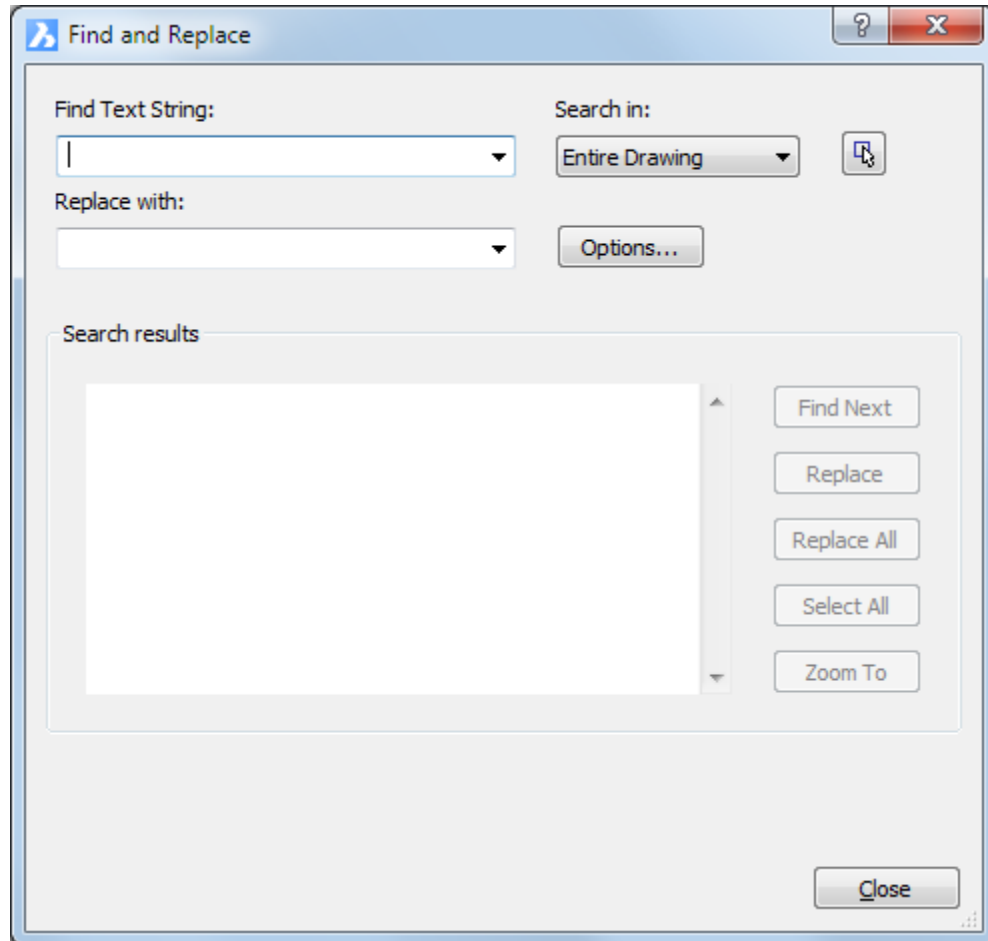
Command: FIND

The *FIND* command lets you specify the text you want to find. You can then find or replace the text. Optionally you can zoom in on the various instances of the text entities that contain the search text.

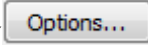
To open the Find and Replace dialog

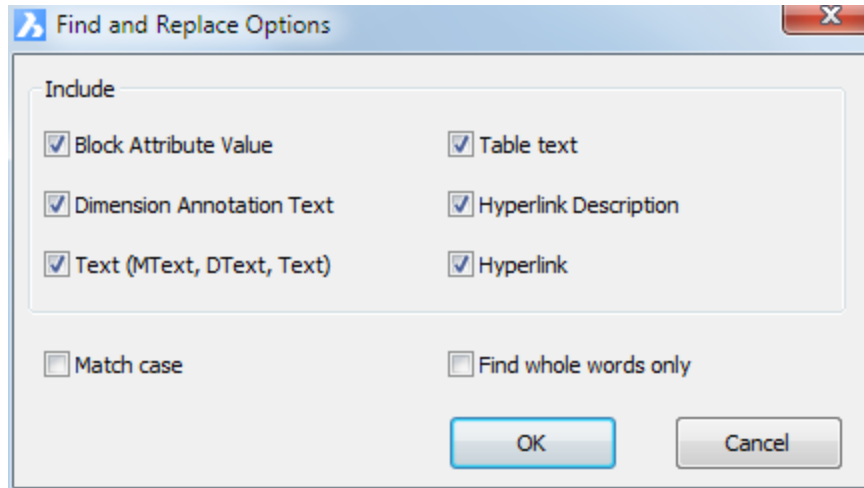
Do one of the following

- Choose *Find...* in the *Edit* menu.
- Type *find* in the command bar, then press Enter.




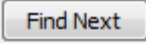
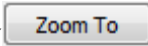
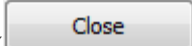
To set the Find and Replace options

1. Open the Find and Replace dialog.
2. Click the *Options* button ().





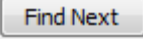
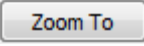

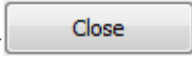
3. Set the *Find and Replace* options.
4. Click the *OK* button.

To find text

1. Open the Find and Replace dialog.
2. (option) Set the Find and Replace options.
3. Type the text in the *Find Text String* field.
4. (option) Click the *Select Entities* button () to compose a selection set.
 - The *Find and Replace* dialog temporarily closes to let you select entities.
 - Right click to conclude the selection of entities.
The *Search in* field reads *Current Selection* instead of *Entire Drawing*.
5. Click the *Find Next* button ().
The content of the first text entity that contains the search string displays in the *Search results* pane.
6. (option) Click the *Zoom To* button ().
The drawing is zoomed in on the text entity.
7. Do one of the following.
 - Repeat steps 7 and 8 to find more instances of the same search string.
 - Repeat steps 2 through 8 to find another text string.
 - Press the Esc key or click the *Close* button () to stop.

NOTE If a selection is active when you launch the *FIND* command, only text entities in the selection set are searched.

To replace text

1. Open the Find and Replace dialog.
2. (option) Set the Find and Replace options.
3. Type the search text in the *Find Text String* field.
4. Type the replacement text in the *Replace with:* field.
5. (option) Click the *Select Entities* button () to compose a selection set.
 - The *Find and Replace* dialog temporarily closes to let you select entities.
 - Right click to conclude the selection of entities.
The *Search in* field reads *Current Selection* instead of *Entire Drawing*.
6. (option) Click the *Replace All* button ().
All instances of the search text are replaced by the replacement text.
7. (option) Click the *Find Next* button ().
The content of the first text entity that contains the search text displays in the *Search results* pane.
 - (option) Click the *Zoom To* button ().
The drawing is zoomed in on the text entity.
 - Do one of the following:
 - Click the *Replace* button ().
 - Repeat this step to find another text entity that contains the search string.
 - (option) Repeat this step until no more occurrences of the search text are found in the drawing or selection set.
8. Do one of the following
 - Repeat steps 2 through 7.
 - Press the Esc key or click the *Close* button () to stop.

NOTE If a selection is active when you launch the *FIND* command, only text entities in the selection set are searched.

Checking Spelling

Command: SPELL

The spelling checker checks the spelling of text in the current drawing. You can check the spelling of one or more selected text entities or check the entire drawing.

During the spelling check, the *Spell* command matches the words in the drawing or the current selection set to the words in the current main dictionary and the current custom dictionary. If a word is not found in either dictionaries you can select the correct spelling in the suggestions field or you can add the word to the current custom dictionary. Custom dictionaries are used for discipline-specific words, such as medical or mechanical.

Dictionary settings

Which dictionaries are used by the *Spell* command is defined by the *DCTMAIN* (main dictionary) and *DCTCUST* (custom dictionary) system variables.

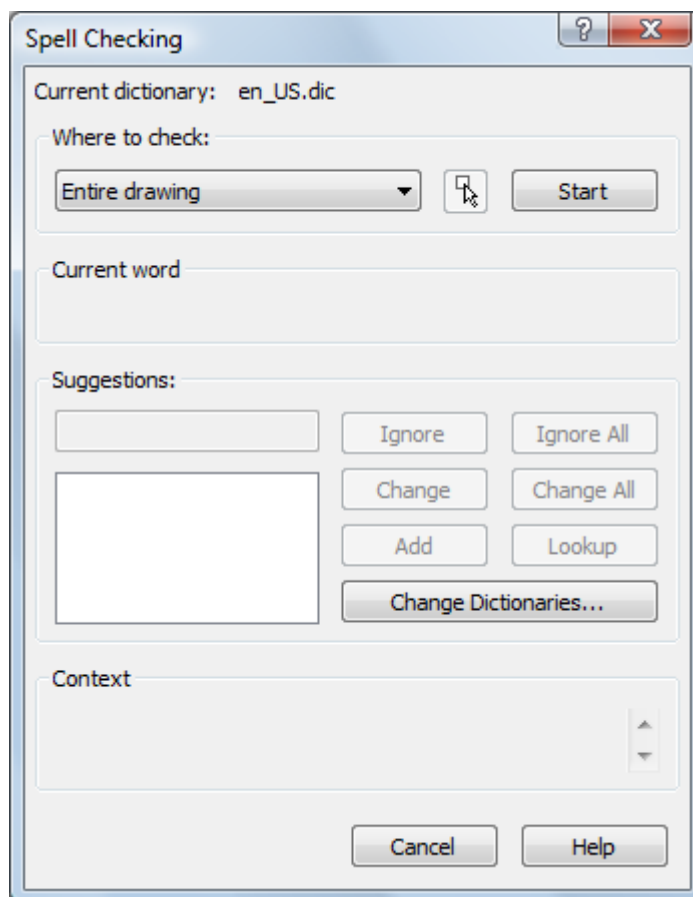
Dictionaries	
Custom spelling dictionary	C:\Users\Louis\AppData\Roaming\Bricsys\Bricscad\V9\en_US\support\Sample.cus
Main spelling dictionary	en_US.dic

NOTE We only deliver the English dictionary because of the license type on the dictionary files : the English version can be distributed freely ; all other languages are not free. However, you can download other languages and use them for free (see To add a main dictionary).


To open the Spell Checking dialog

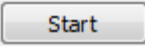
Do one of the following:

- Choose *Check Spelling* in the *Tools* menu.
- Type *spell* in the command bar, then press Enter.

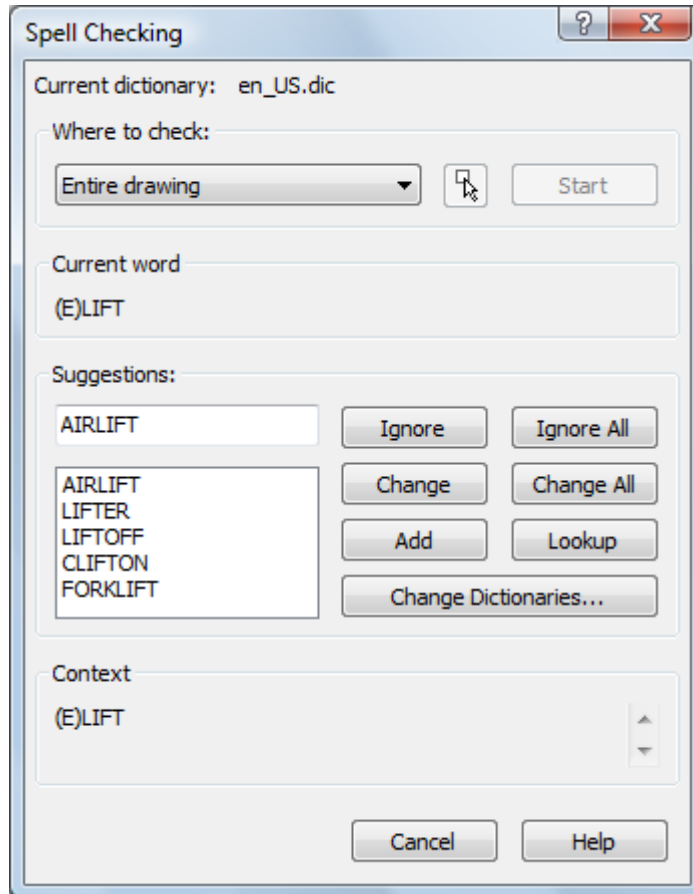




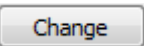
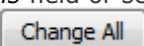

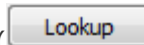
To check the spelling

1. Open the Spell Checking dialog.
2. (option) Click the *Select* button () to compose a selection set.

3. (option) Change the current dictionaries.
4. Click the *Start* button ().

The first possibly misspelled word displays in the *Current word* section.
The content of the text entity in which the *Current word* is found displays in the *Context* section.

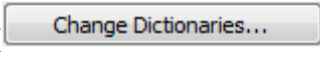


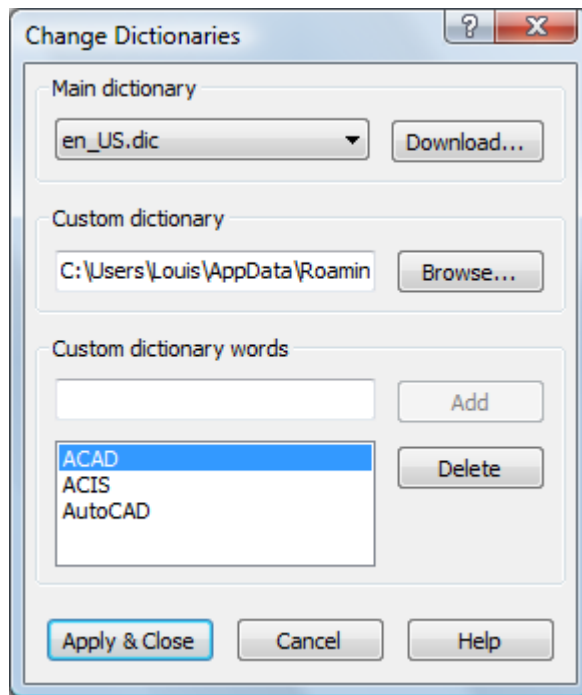
5. Do one of the following
 - Click the *Ignore* button () to leave this instance of the *Current word* unchanged.
 - Click the *Ignore All* button () to leave all instances of the *Current word* unchanged.
 - Accept the word in the *Suggestions* field or select a word in the *Suggestions* list, then click the *Change* button () to replace this instance of the *Current word* by the word in the *Suggestions* field.
 - Accept the word in the *Suggestions* field or select a word in the *Suggestions* list, then click the *Change All* button () to replace all instances of the *Current word* by the word in the *Suggestions* field.
 - Click the *Add* button () to add the *Current word* to the current custom dictionary.
 - Click the *Lookup* button () to list words similar to the word in the *Suggestions* field.
6. Repeat step 5 until the *Spelling check is complete* message displays.

NOTE If a selection is active when you launch the *Spell* command, only text entities in the selection set are checked.

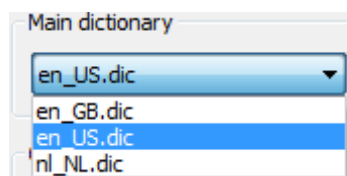
To change dictionaries

1. Open the Spell Checking dialog.

2. Click the *Change Dictionaries...* button ().
The *Change Dictionaries* dialog opens.



3. Click the *Main dictionary* list button to change the main dictionary.

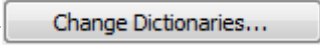



4. Select a new main dictionary.
5. (option) Click the *Browse...* button to select a new *Custom dictionary*.
6. (option) Type a word in the *Custom dictionary words* field, then click the *Add* button to add a word to the current custom dictionary.
7. Click the *Apply & Close* button.

NOTE Custom dictionaries are best saved in the *Support* folder of the Roamable root folder.
If you want to share the custom dictionary with MS Word, point to the MS Word custom dictionary in step 5 in the above procedure. By default, *CUSTOM.DIC* is the custom dictionary in MS Word, which can be found under:
C:\Documents and Settings\<user name>\Application Data\Microsoft\Proof\CUSTOM.DIC on a Windows XP computer
C:\Users\<user name>\AppData\Roaming\Microsoft\Proof\CUSTOM.DIC on a Windows Vista (or higher) computer.

To add a new main dictionary

1. Open the Spell Checking dialog.

2. Click the *Change Dictionaries...* button ().

3. Click the *Download...* button ().


Your browser opens: <http://wiki.services.openoffice.org/wiki/Dictionaryes>

4. Do one of the following:
 - If necessary, click show next to Contents to display the list of available dictionaries.
Click the language you want to download the dictionary for.
You are prompted to *Save* or *Open* a ZIP file (e.g. En_GB.zip), containing the dictionary files.



- Go to the *Extensions Repository* to get the new dictionaries.



IMPORTANT NOTE: From OpenOffice.org 3.0 on the dictionary wizard is no longer available -- All new dictionaries are now available via [the Extensions Repository](#) .

Scroll down to the language of your choice, then click the link to open the page of the selected language.

Click the *Get it!* button to start downloading the *.oxf file.

Get it!

Operating System: System Independent

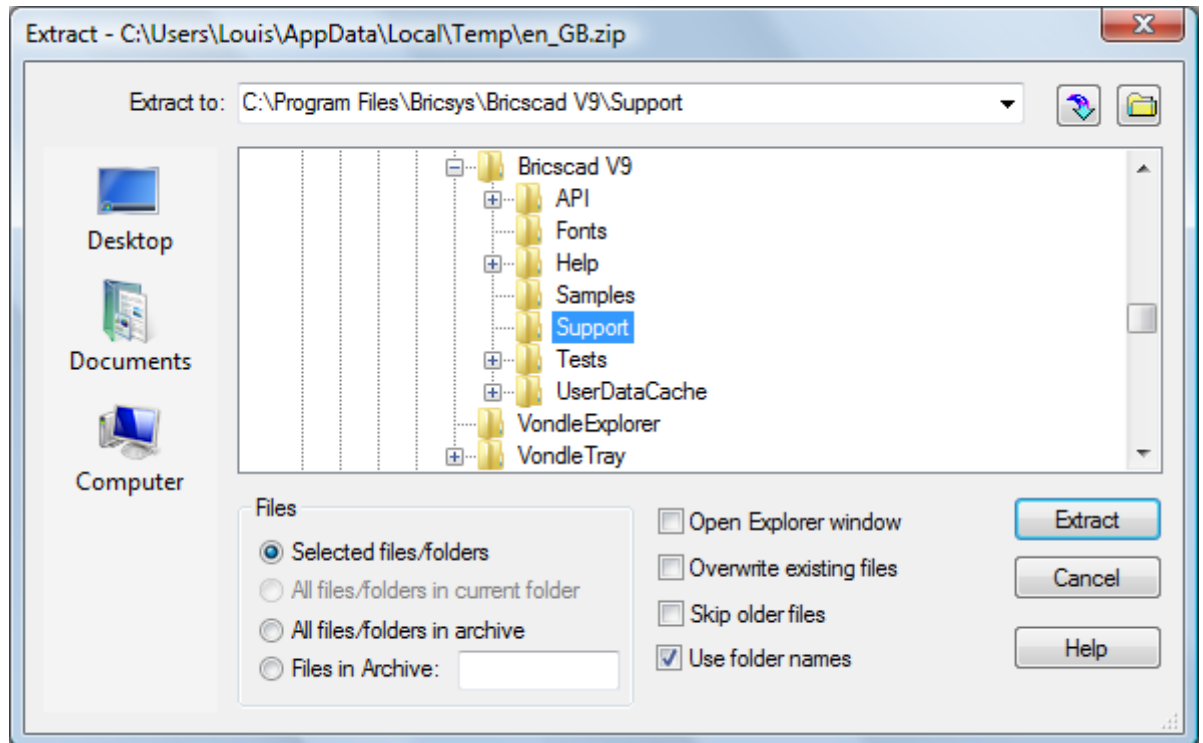
Compatible with: OpenOffice.org 3.2 or higher.

Version: 4.1 **Date:** 2011-Mar-27 **Size:** 2.1 MB

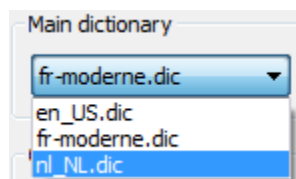
[View all releases](#)

Change the *ext* extension to *zip*.

5. Extract the *.*aff* and *.*dic* files in the ZIP file to the *C:\Program Files\Bricsys\BricsCAD Vx\Support* folder.



The dictionary is now available in the *Main Dictionary* list on the *Change Dictionaries* dialog box.



To add a new custom dictionary

1. Open a text editor, such as *Notepad*.
2. (option) Type the words you want to use in your custom dictionary. Each word must be on a separate line.
3. Save the file to the *Support* folder of the Roamable root folder. Make sure the file extension is *.cus*.

Working with Fields

Commands: FIELD, UPDATEFIELD, MTEXT and TABLE

A field is a variable text that. When a field is updated it displays the latest value of the data source it refers to. Fields can be used to insert drawing properties, custom properties or object properties in multiline texts or tables or as an attribute in a block. Fields in attributes are especially useful in title blocks: using drawing, sheet set, subset and sheet properties title blocks can be filled out automatically.

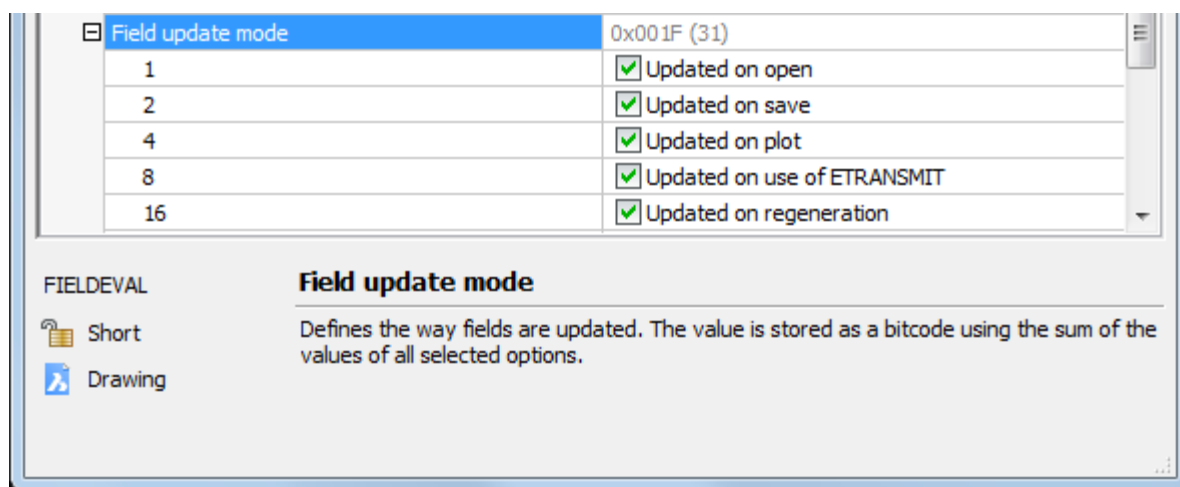
See the DWGPROPS command to learn more about default and custom drawing properties.

See the SHEETSET command to learn more about default and custom sheet set, subset and sheet properties.

If no value is available, a field displays hyphens (----).

If the FIELDDISPLAY system variable is ON, fields display on a light gray background which does not print.

The FIELDDEVAL system variable controls when fields are updated.

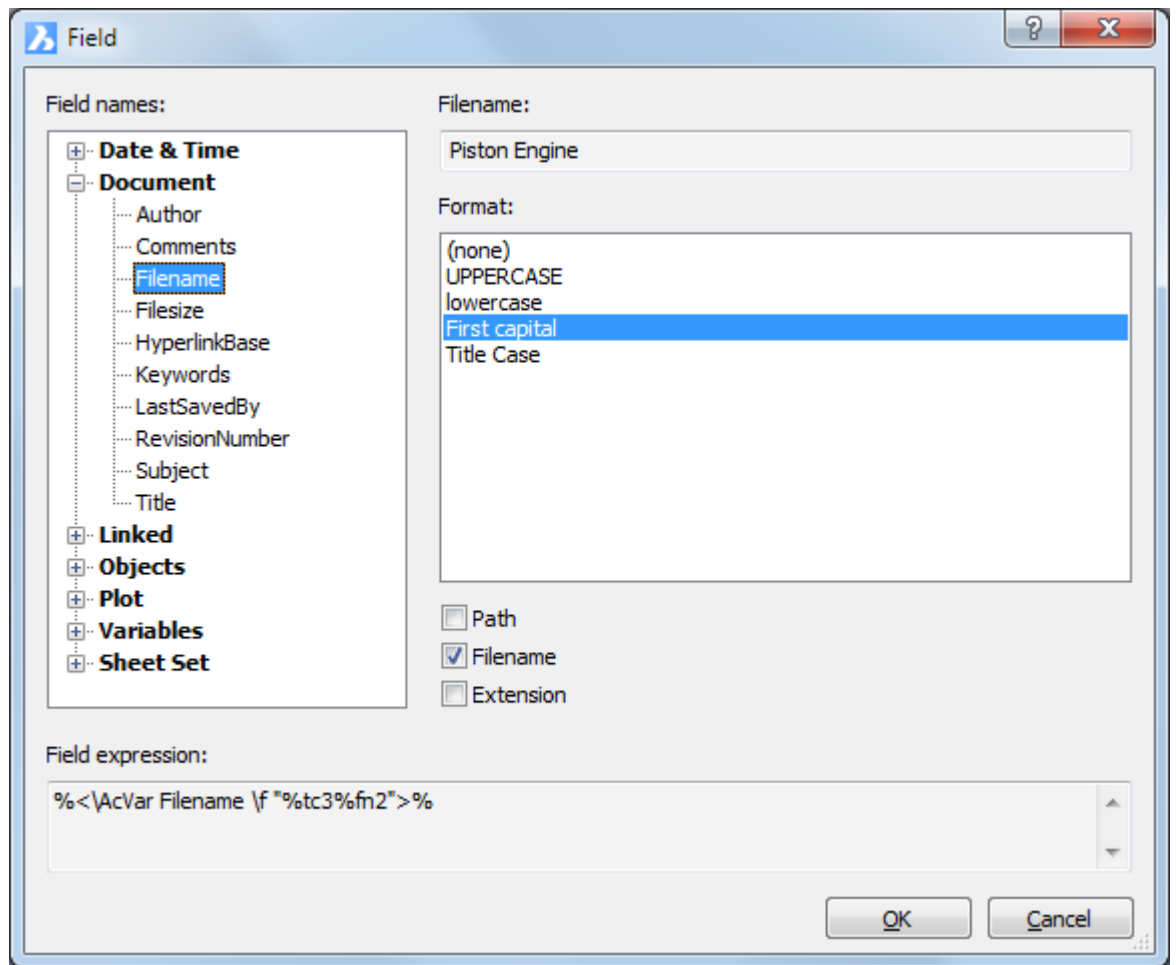


General procedure to create fields

1. Do one of the following:

- Click the *Insert Field* tool button (📄) on the *Insert* toolbar.
- Choose *Insert Field...* in the *Insert* menu.
- Type *field* in the command bar, then press Enter.
- Click the *Field* tool button (📄) on the *Text Formatting* toolbar when creating or editing Mtext or a table cell content.

The *Field* dialog box displays.



2. Under *Field Names*, expand a group, then choose a field name.
The settings for the selected field display.
3. Adjust the settings, then click the *OK* button.
The *Field* dialog box closes.

The field is created in the Mtext or table cell being edited

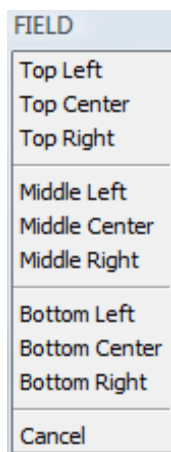
or

The command bar reads: Specify start point or [Height/Justify]:
A context menu displays:




- (option) Type *H* or choose *Height* in the context menu to define the text height.

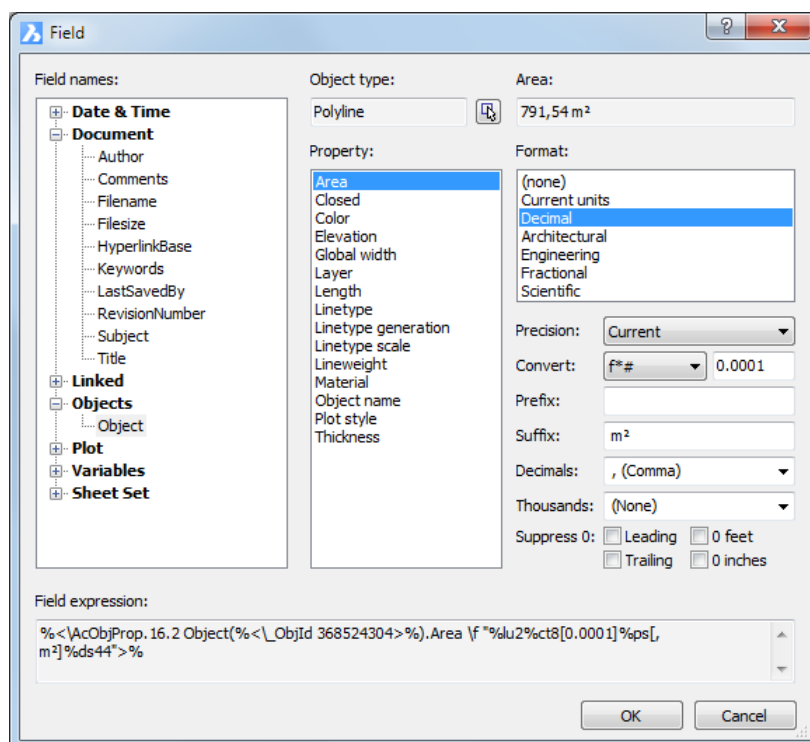
- (option- Type *J* or choose *Justify* in the context menu to set the text justification.



- Specify the insertion point of the field in the drawing.
An *Mtext* entity, containing the field is created.

To display an entity property in a field

1. Launch the Field command.
2. Expand *Objects* in the *Field names* list and select *Object*.
3. Click the *Select Entity* button () on the *Field* dialog box.
The *Field* dialog box closes temporarily.
The command bar reads: Select Entity:
4. Click the entity you want to display a property of.
The *Field* dialog box displays again.
5. Select the entity property in the *Property* list.



6. Set the display layout:
 - *Format*: choose a unit format.
 - *Precision*: select a precision in the list button


- *Convert*: this option allows to recalculate the field value.
 - Choose a formula in the list; e.g. $f \times \#$, where f is the factor you type in the Convert field and $\#$ is the field value.
 - Type a factor in the Convert field; e.g. 0.0001 to express the area of a polyline in square meters in a cm drawing.
 - *Prefix*: type a text string
 - *Suffix*: type a text string
 - *Decimals*: select a decimal separator character
 - *Thousands*: select a thousands separator character
 - *Suppress 0*: allows to suppress leading and/or trailing zeros
7. Click the *OK* button.
 8. Specify a point to place the field in the drawing.

To edit a field

1. Edit the table cell or Mtext entity that contains the field.
2. Do one of the following:
 - Double click the field.
 - Right click the field, then choose *Edit Field ...* in the context menu.
 - Select the field, then press *Ctrl + F*.

The *Field* dialog box displays.
3. Adjust the properties of the field.
4. Click the *OK* button to apply the changes.

To update a field

1. Do one of the following:
 - Click the *Update Field* tool button () on the *Redraw / Regen* toolbar.
 - Choose *Update Fields...* in the *Tools* menu.
 - Type *updatefield* in the command bar.

The command bar reads: Select field(s) to update:
2. Select the field(s) you want to update, then right click or press Enter.
The selected fields are updated.

To convert a field to text

3. Edit the table cell or Mtext entity that contains the field.
4. Select the field.
5. Right click, then choose *Convert Field to Text* in the context menu.

Tables

Commands: TABLE, -TABLE, TABLEDIT, TABLEEXPORT, TABLEMOD, TABLESTYLE

A table is an entity that is composed of rows and columns, much like a Microsoft Excel spreadsheet. A table entity can be created as an empty table or you can import data from a CSV (**C**omma **S**eparated **V**alues) file. The appearance of the table text and gridlines is controlled with a table style. Table styles are saved in the DWG file.

A default table consists of a *Title* cell (A), a row of *Header* cells (B) and one or more rows of *Data* cells (C).

A table can have one of two directions: *Down* (title row at the top) or *Up* (title row at the bottom).


A				
B	B	B	B	B
C	C	C	C	C
C	C	C	C	C
C	C	C	C	C

C	C	C	C	C
C	C	C	C	C
C	C	C	C	C
B	B	B	B	B
A				

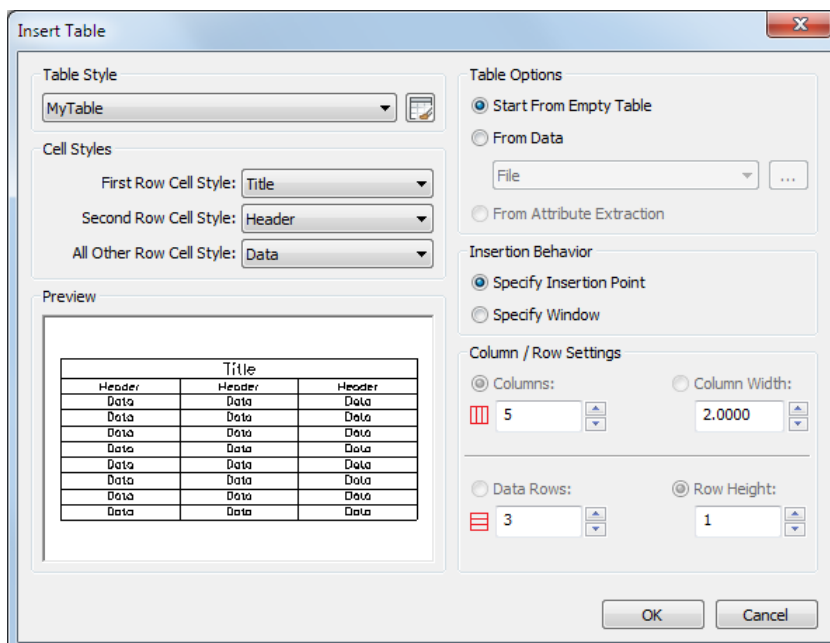
Table with a Down direction (left) and a Up direction (right).

To create an empty table

1. Do one of the following:

- Click the *Table* tool button () on the *Draw* toolbar.
- Choose *Insert Table...* in the *Draw* menu.
- Type *table* in the command bar, then press Enter.

The *Insert Table* dialog displays:



The *Insert Table* dialog box is shown with the following settings:



- Table Style:** MyTable
- Cell Styles:**
 - First Row Cell Style: Title
 - Second Row Cell Style: Header
 - All Other Row Cell Style: Data
- Table Options:**
 - ☒ Start From Empty Table
 - ☐ From Data (File:)
 - ☐ From Attribute Extraction
- Insertion Behavior:**
 - ☒ Specify Insertion Point
 - ☐ Specify Window
- Column / Row Settings:**
 - ☒ Columns: 5
 - ☐ Column Width: 2.0000
 - ☐ Data Rows: 3
 - ☒ Row Height: 1

The **Preview** section shows a table with 5 columns and 3 data rows. The first row is labeled 'Title', the second row is labeled 'Header', and the remaining three rows are labeled 'Data'.

2. Choose a Table Style from the *Table Style* list.
3. (option) Set the cell style for the first, second and other rows.
By default the cell style for the first row is *Title*, the cell style for the second row is *Header* and the cell style for the other rows is *Data*.
4. Under *Table Options*, choose *Start from Empty Table*.
5. Under *Insertion Behavior* do one of the following:
 - Choose *Specify Insertion Point*, then specify the number of columns and the column width and number of data rows and the row height.
 - Choose *Specify Window*, then do one of the following:
 - Specify the number of columns and data rows.
 - Specify the column width and the row height.
 - Specify the column width and the number of data rows.
 - Specify the number of columns and the row height.
6. Click the *OK* button.
Depending on your choice in step 5 you are prompted to specify an insertion point or a window.
The table is created and the *Text Formatting* toolbar displays (see Create multiline text).
7. Do one of the following:
 - Start adding text in the table (see Edit the table content).
 - Click outside the table to close the *Text Formatting* toolbar.

To create a table from data

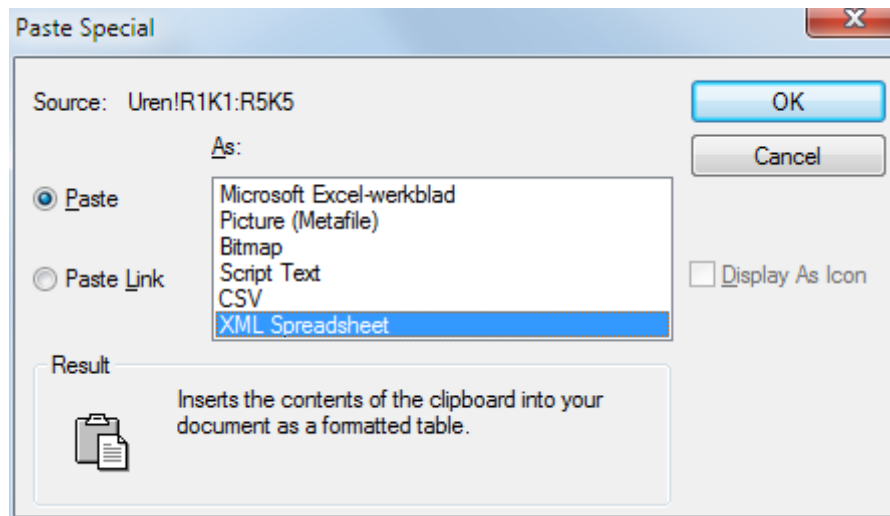
When importing data from a CSV (**C**omma **S**eparated **V**alues) file, please make sure the list separator character on your system is the same as the one used in the CSV file (see Defining the list separator character).

1. Do one of the following:
 - Click the *Table* tool button () on the *Draw* toolbar.
 - Choose *Insert Table...* in the *Draw* menu.
 - Type *table* in the command bar, then press Enter.
 The *Insert Table* dialog displays.
2. Choose a Table Style from the *Table Style* list.
3. Under *Table Options*, choose *From Data*.
4. Click the Browse button ()
The *Open* dialog displays.
5. Choose a CSV file, then press the *Open* button.
6. Click the *OK* button on the *Insert Table* dialog.
The dialog box closes and the table is attached to the cursor.
7. Click a point in the drawing to place the table.
The table is created and the *Text Formatting* toolbar displays (see Create multiline text).
8. Do one of the following:
 - Start editing the text in the table (see Edit the table content).
 - Click outside the table to close the *Text Formatting* toolbar.
9. (option) Select the table and drag the grips to adjust the dimensions of the table.

To create a table from an Excel spreadsheet

1. Open the spreadsheet in *Microsoft Office Excel*.
2. Select the cells you want to create a table from.
3. Copy the selection to the clipboard:
 - Press *Ctrl + C*.
 - Right click and choose *Copy* in the context menu.

4. In BricsCAD, choose *Paste special...* in the *Edit* menu.
The *Paste Special* dialog box displays:

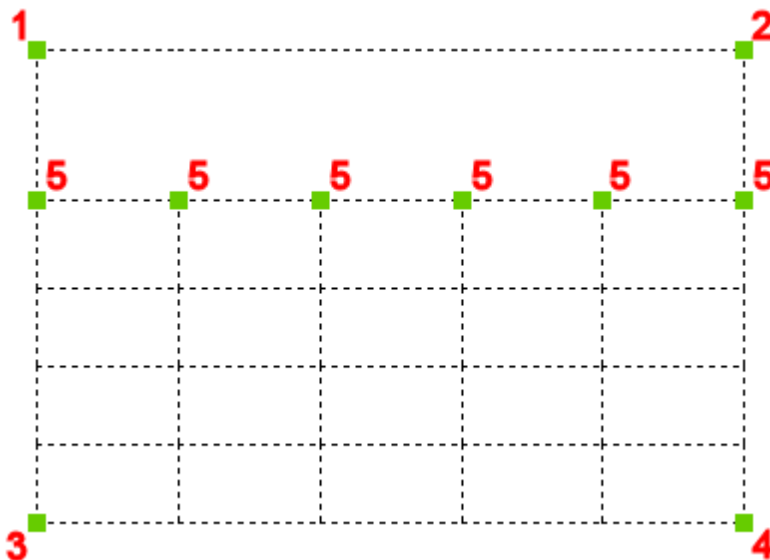


5. Choose *XML Spreadsheet* in the *As* option list.
6. Click in the drawing to insert the table.

Editing a table using grips

When a table is selected, multiple grips display:

1. Moves the table.
2. Modifies the width uniformly.
3. Modifies the height uniformly.
4. Modifies the width and height uniformly.
5. Modifies the width of a column.



Editing a table using the Table toolbar

1. To open the *Table* toolbar, do one of the following:
 - Click the cell you want to edit.
 - Press and hold the left mouse button then define a rectangle to select multiple cells
 Column letters and row numbers display.
The column(s) and row(s) of the selection highlight.

A rectangle with four handles is drawn around the selection.

Table					
	A	B	C	D	E
1	A				
2	B	B	B	B	B
3	C	C	C	C	C
4	C	C	C	C	C
5	C	C	C	C	C








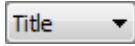

2. Do one of the following:
 - Click a cell to select it.
 - Press and hold the left mouse button, then drag the cursor to select multiple cells.
3. (option) Drag the grips of the selection to modify the dimensions.
4. (option) Click a tool in the Table toolbar.
5. (option) Right click to display a context menu:

Delete Cell Content
Clear Cell Overrides
Clear Table Overrides

- *Delete Cell Content*: deletes the text in the selected cells.
- *Clear Cell Overrides*: resets all cell properties of the selected cells to the Cell Style settings as defined in the Table Style.
- *Clear Table Overrides*: resets all cell properties of all cells to the Cell Style settings as defined in the Table Style.

Table tools

Icon	Tool name	Description
	Insert row above	Inserts an empty row above the selection.
	Insert row below	Inserts an empty row below the selection.
	Delete row	Deletes the row(s) of the selection.
	Insert column left	Inserts an empty column at the left of the selection.
	Insert column right	Inserts an empty column at the right of the selection.

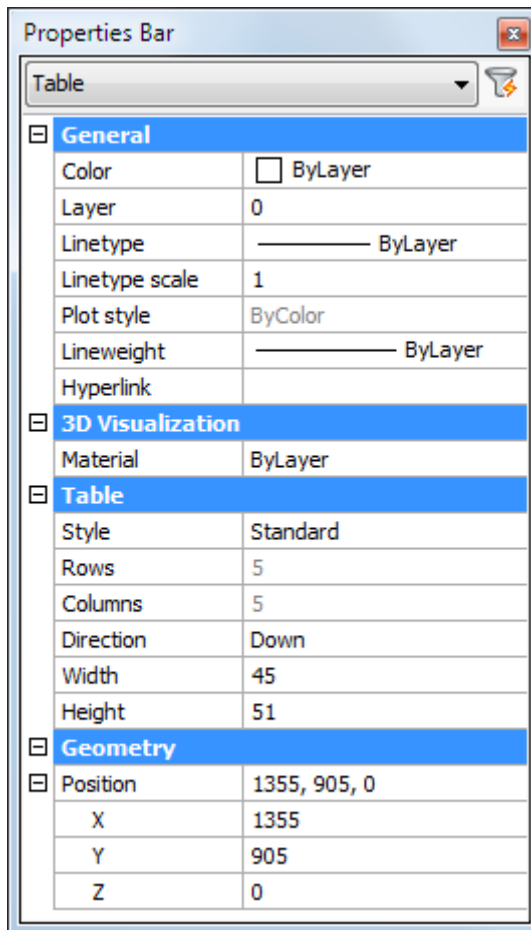
	Delete column	Deletes the column(s) of the selection.
	Merge	Merges the selected cells.
	Unmerge	Splits previously merged cells.
	Border style	Sets the border style of the selection (not implemented yet).
	Text alignment	Sets the text alignment of the selection.
	Lock / Unlock	Locks / Unlocks the selection.
	Set cell format	Sets the cell format of the selection (not implemented yet)
	Set cell style	Sets the style of the selected cells: <i>Title</i> , <i>Header</i> or <i>Data</i> .
	Set cell background	Sets the background color of the selection.

6. (option) Repeat steps 3 and 4 to continue editing the table.
7. (option) Double click a cell to edit the cell text.
8. (option) Do one of the following:
 - Click another cell.
 - Press and hold the left mouse button, then drag the cursor to select multiple cells.
9. Click outside the table to close the *Table* toolbar.

Editing a table in the Properties bar

1. Click one of the grid lines.

The properties of the table display in the Properties bar.

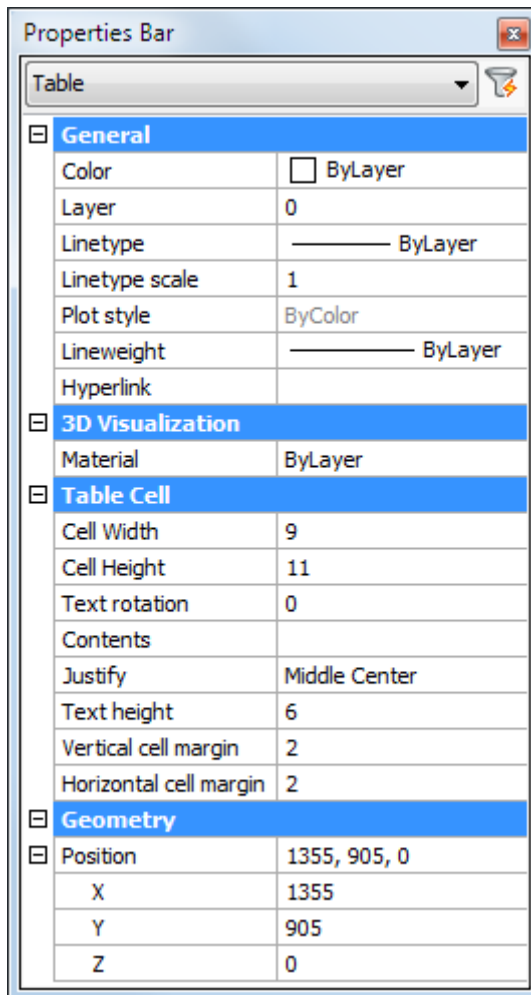


2. (option) Apply a different Table style.
3. (option) Change the table direction.
4. (option) Modify the width and/or height of the table.
5. (option) Modify the vertical and/or horizontal cell margins.
6. Press the Esc key to stop editing the table.

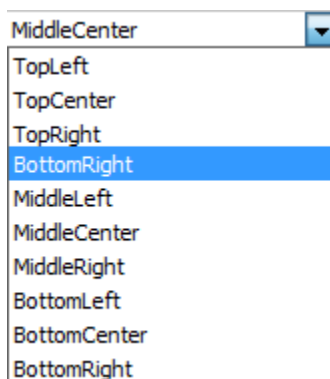
Editing a cell in the Properties bar

1. Click the cell.

The properties of the cell display in the Properties bar.



2. (option) Modify the width and/or height of the cell.
The width and height of the column and/or row of the cell are changed.
3. (option) Modify the text rotation.
The options are: 0°, 90°, 180° and 270°.
4. (option) Edit the cell content.
5. (option) Choose a different text alignment option.



To edit the table content

1. Do one of the following:
 - Double click the cell you want to edit.
 - Type *tableedit* in the command bar, then press Enter and click the cell you want to edit.

The *Text Formatting* toolbar displays above the selected cell (see Create multiline text).
2. Edit the text in the cell.
(option) Press Alt + Enter to create a second line in the same cell.
3. To edit another cell:
 - Press the TAB key to go to the next cell in the same row.
 - Press Shift + TAB to go to the previous cell in the same row.
 - Press Enter to go the next cell in the same column.
 - Press Shift + Enter to go to the previous cell in the same column.
 - Press one of the Arrow keys.
4. Click to stop editing the table content.

To export a table

When exporting data to a CSV (**C**omma **S**eparated **V**alues) file, the list separator character off your system is used in the CSV file (see Defining the list separator character).

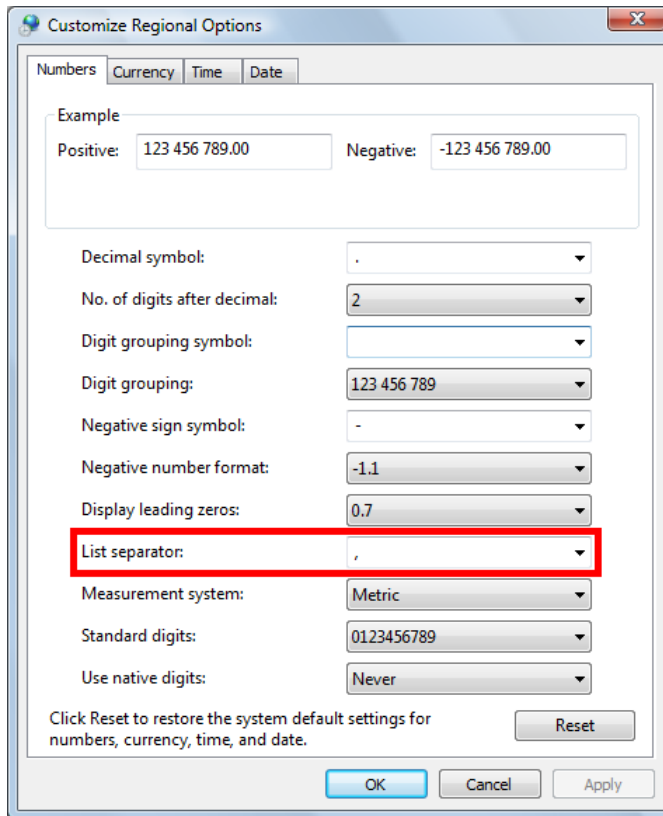
1. Type *tableexport* in the command bar, then press Enter.
The command bar reads: Select table to export:
2. Click the table you want to export the data of.
The Export Data dialog opens.
3. Select a folder to save the data in.
4. Type a name in the *File name* field.
5. Click the *Save* button.
The data in the selected table are saved as a CSV (**C**omma **S**eparated **V**alues) file.

Defining the list separator character

1. Click the Start button in the Windows Taskbar, then choose Control Panel.
2. Do one of the following:
 - On the Control Panel Home window, choose: Clock, Language and Region > Regional and Language Options.
 - On the Control Panel Classic View window, double click the Regional and Language Options icon.

The Regional and Language Options dialog box displays.
3. On the Regional and Language Options dialog box click the Customize this format... button.

The Customize Regional Options dialog box displays:



4. On the Numbers page of the Customize Regional Options dialog box, go to List separator, then click the down arrow and select either comma (,) or semicolon (;).
5. Click the OK button on the Customize Regional Options and Regional and Language Options dialog boxes.

Table Styles

Command: TABLESTYLE

The appearance of a table is controlled by a table style. Each drawing contains the *Standard* table style, which cannot be deleted. You can modify the *Standard* table style to fit your needs or create your own table styles. Table styles are created and edited in the *Table Styles* category of the Drawing Explorer dialog.

In a table there are 3 cell types: *Data*, *Header* and *Title*. Each cell type has its own cell style settings. By default all cells in a row share the same cell style settings, but you can edit the settings of each cell afterwards.

To open the Table Styles Explorer

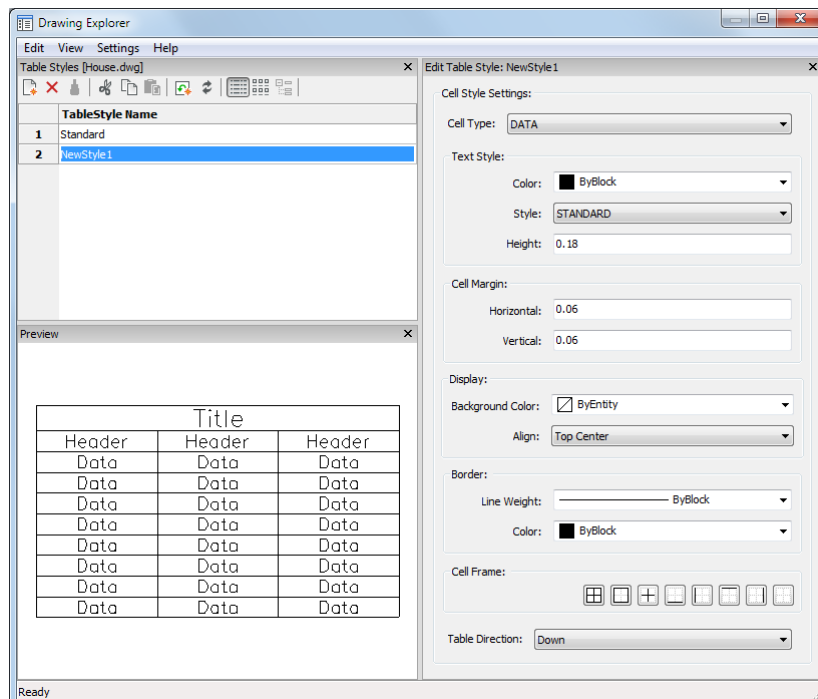
Do one of the following:

- Choose *Drawing Explorer > Table Styles...* in the *Tools* menu.
- Type *tablestyle* in the command bar, then press Enter
- Click the *Explore Table Styles* tool button (📄) on the Insert Table dialog.

To create a table style

1. Open the Drawing Explorer - Table Styles dialog displays.
2. In the *Drawing Explorer - Table Styles* dialog do one of the following:
 - Choose *New* in the *Edit* menu.
 - Click the *New* tool button (📄).

A new style is created using default settings.



3. Rename the style, replacing the *NewStyle1* default name.
4. Select a cell type: *Data*, *Header* or *Title*.
5. Define the cell style settings for the selected cell type.
6. Repeat the previous steps for each cell type.

To edit a table style

1. Open the Drawing Explorer - Table Styles dialog displays.
2. In the *Details* pane of the *Drawing Explorer - Table Styles* dialog, select the table style you want to edit.
3. Select a cell type: *Data*, *Header* or *Title*.
4. Edit the cell style settings for the selected cell type.
5. (option) Repeat the previous steps for another cell type.

To apply a style to a table

See To edit a table using the Properties bar.

To delete a table style






1. Open the Drawing Explorer - Table Styles dialog displays.
2. Select the table style you want to delete.
Press and hold the Ctrl key to select multiple table styles.
3. Do one of the following:
 - Right click, then choose *Delete* in the context menu.
 - Click the *Delete* button in the *Details* menu on the *Drawing Explorer - Table Styles* dialog

NOTE It is not possible to delete a table style that is being used in the drawing.

Modifying Commands Overview

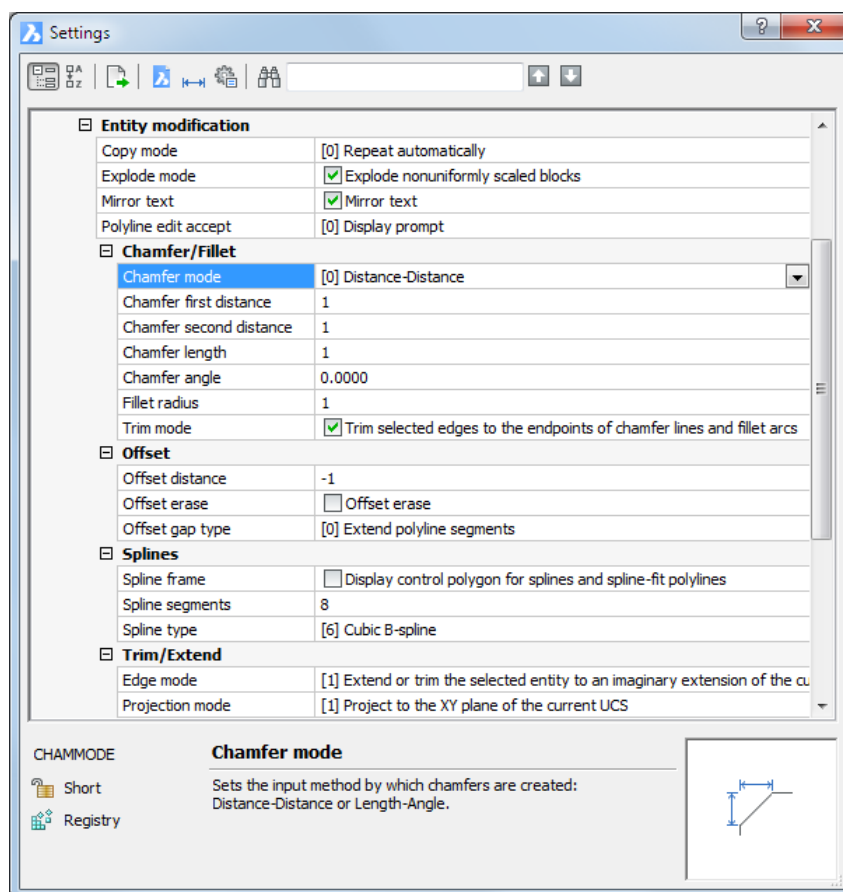
All entity modification tools can be found either on the *Modify* toolbar or in the *Modify* menu.

Icon	Tool name	Command	Description
	Move	MOVE	Moves entities to another location in the same drawing or into another drawing.
	Copy	COPY	Draws a duplicate or multiple duplicates of the selected entities.
	Offset	OFFSET	Creates a parallel or offset copy of curves and lines.
	Scale	SCALE	Changes the scale of existing entities, either enlarging them or reducing them proportionately in x, y, and z directions.
	Rotate	ROTATE	Rotates entities around a specified point.
	3D Rotate	ROTATE3D	Rotates entities around a three-dimensional axis.
	Mirror	MIRROR	Moves or copies the reflected image of entities about a line.
	3D Mirror	MIRROR3D	Moves or copies the reflected image of entities about a plane.
	Array	ARRAY	Creates multiple copies of entities in one of two symmetrical patterns: rectangular (rows and columns) or polar (circular).
	3D Array	3DARRAY	Creates multiple copies of entities in three dimensions. Entities are arrayed in a three-dimensional rectangular (rows, columns and levels) pattern or a two-dimensional polar (circular) pattern in three-dimensional space.
	Break	BREAK	Splits an entity into two entities.
	Join	JOIN	Joins two entities (two or more lines or two or more arcs) into one entity.
	Trim	TRIM	Erases the portions of selected entities that cross a specified boundary.
	Align	ALIGN	Aligns selected entities with other entities in three-dimensional space.
	Extend	EXTEND	Lengthens lines, arcs, polylines or rays to meet another entity.
	Edit Length	LENGTHEN	Changes the length of a line, polyline, freehand entity or arc.
	Stretch	STRETCH	Moves a portion of a drawing while retaining connections to other parts of the drawing.
	Measure	MEASURE	Divides a selected entity into segments by placing markers (points or blocks) at specified intervals along its length or circumference.
	Divide	DIVIDE	Places markers (points or blocks) along a selected entity. The markers evenly divide the entity into the specified number of equal parts.
	Chamfer	CHAMFER	Creates a chamfer, or a beveled edge, at the intersection of two 3D solids, lines, rays, or infinite lines.
	Fillet	FILLET	Creates a fillet, or rounded corner, at the intersection of 3D solids, two lines, rays, or infinite lines.

	Edit Polyline	PEDIT	Edits a two-dimensional or three-dimensional polyline, or a polygon mesh.
	Explode	EXPLODE	Ungroups a block, polyline, polyface mesh, solid, or hatch, creating separate entities for each element.
	Region	REGION	Converts a closed entity into a two-dimensional region.
	Change	CHANGE	Changes the location, size, color, elevation, layer, linetype, linetype scale, lineweight, and three-dimensional thickness of entities.
	Properties	PROPERTIES	Opens the BricsCAD Properties Bar (if not already open).

Entity Modification Settings

Entity modification settings are found in the *Entity modification* settings group of the *Drawing* settings category in the Settings dialog.




Adjusting the Draw Order

Commands: DRAWORDER, HATCHTOBACK, TEXTTOFRONT

If new entities overlap existing entities, they will display and print on top of the previously drawn entities. The *Draw Order* tool can change the order in which overlapping entities are displayed and printed. You can move entities to the front or the back, or above or under another entity.

Using Draw Order

1. Do one of the following:

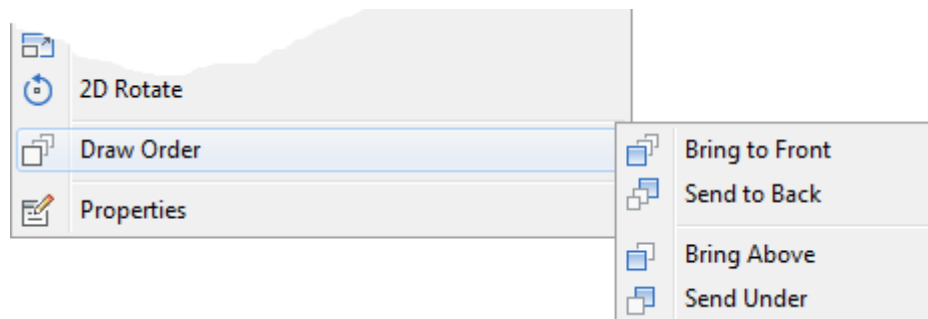
- Click the *Draw Order* tool button () on the *Draw Order* toolbar.
- Choose *Draw Order* in the *Tools* menu.
- Type *draworder* in the command bar.







The command bar reads: Select entities to change the draw order:

2. Select the entities, then right click or press enter.
The command bar reads: Change draw order: Above/Under/Clear/Front/<Back>:
3. Choose the appropriate draw order tool option.
4. Depending on your choice:
- If you have chosen *Above* or *Under*, the command bar reads: Select reference entity:
Identify the reference entity. The display of the selection set is changed with respect to the reference entity.
 - If you have chosen *Front* or *Back*, the display of the selection set is changed accordingly.

Using Draw Order tools

1. Select the entities you want to change the draw order of.
2. Do one of the following.
- Click the appropriate tool button on the *Draw Order* toolbar.
 - Right click, then choose *Draw Order* in the context menu and select the appropriate tool.



Tool Button	Tool Name	Description
	Bring To Front	Places the selection set on top.
	Bring Above	Places the selection set above the reference entity. (*)
	Send Under	Places the selection set under the reference entity. (*)
	Send to Back	Moves the selection set to the back.
	Hatch to Back	Sends all hatch entities to the back.
	Text and Dimensions to Front	Brings all texts and/or dimension entities to front. Options: <i>Text</i> / <i>Dimensions</i> / <i>Both</i> can be chosen either in the command bar or in the prompt menu. The default option is <i>Both</i> .

(*) You are prompted to select the reference entity.

Grips editing



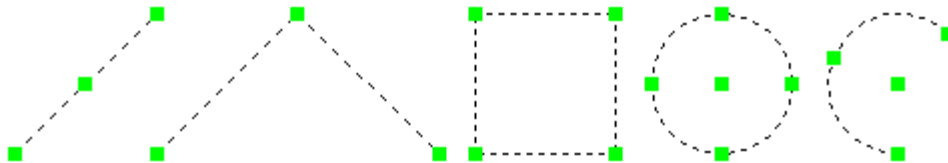
Grips Editing

Grips display as small colored squares at key-points on selected entities.

Whether grips display or not and the appearance of grips is controlled through a series of grips settings.

When the display of grips is turned on, grips can be dragged using the mouse in order to stretch, move, mirror, rotate or scale entities.

The *Copy* option of the grips editing commands allows to create multiple copies of the entities being edited.



To select a grip, move the mouse over it, then click. Selected grips display in a different color, which is defined by the *HOTGRIP* system variable. A selected grip is called a hot grip. By default hot grips display in red.

The *midpoint* grip on lines and infinite lines, the *startpoint* grip on rays, the *center* grip on circles, arcs, ellipses and elliptical arcs, the grip on points and the *insertion point* grip of texts, mtexts, blocks and Xrefs can be used to quickly move such entities.

NOTE When DRAGSNAP is ON, dragged entities display at the current entity snap location.

Selecting multiple grips

1. Select all entities you want to edit.
2. Press and hold the Shift key, then click the first grip.
The selected grip displays in red.
3. Keep the Shift key pressed, then click more grips.
All selected grips display in red.



4. Release the Shift key, then click the hot grip you want to use as the base point.
The hot grips move simultaneously.

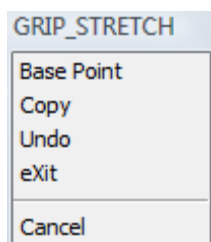
Grip Edit Command Options

Keyboard	Prompt Box	Description
B	Base Point	Prompts you to specify a new base point, other than the selected grip.
C	Copy	Copies the selected entities and keeps the originals. By default, multiple copies can be created. Press Enter to stop making copies.
U	Undo	If the <i>Copy</i> option is chosen, deletes the latest copy.
R	Reference	(Applies to <i>Grip Rotate</i> only) Allows to specify a reference angle. Type the angle in the command bar or specify two points.
X	Exit	Aborts the grip edit command.

To stretch entities using grips

1. Select the entity or entities.
2. Select the grip or grips you want to manipulate.
3. Click the grip you want to use as the base point.
All selected grips move simultaneously.

The *GRIP_STRETCH* prompt box displays:



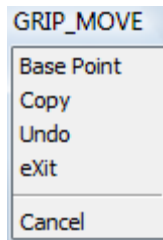
The command bar reads: Enter new point: Base point/Copy/Undo/eXit:

4. Do one of the following to define the displacement point:
 - Click in the drawing.
 - Enter the coordinates, then press Enter.
 - Key-in the distance, then press Enter.
Use ORTHO or Polar Tracking to control the angle.

To move entities using grips

1. Select the entity or entities.
2. Select the grip or grips you want to manipulate.
3. Click the grip you want to use as the base point for the move.
The *Grip Stretch* command is launched.
4. Do one of the following:
 - Right click, then select *Move* in the context menu.
 - Hit the *Space bar* or *Enter* key to cycle through the *Grips Editing* commands until *Move* is active.

The *GRIP_MOVE* prompt box displays:



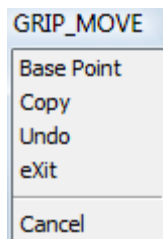
The command bar reads: Enter move point: Base point/Copy/Undo/eXit:

5. Do one of the following to define the displacement point:
 - Click in the drawing.
 - Enter the coordinates, then press Enter.
 - Key-in the distance, then press Enter.
Use ORTHO or Polar Tracking to control the angle.

To copy entities using grips

1. Select the entity or entities.
2. Select the grip or grips you want to manipulate.
3. Click the grip you want to use as the base point for the move.
The *Grip Stretch* command is launched.
4. Do one of the following:
 - Right click, then select *Move* in the context menu.
 - Hit the *Space bar* or *Enter* key to cycle through the *Grips Editing* commands until *Move* is active.

The *GRIP_MOVE* prompt box displays:



The command bar reads: Enter move point: Base point/Copy/Undo/eXit:

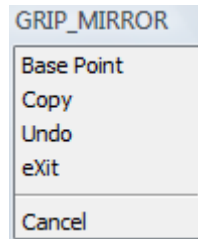
5. Do one of the following
 - Choose *Copy* in the context menu.
 - Type *C*, then hit the *Space bar* or *Enter* key.
6. Do one of the following to define the displacement point:
 - Click in the drawing.
 - Enter the coordinates, then press Enter.
 - Key-in the distance, then press Enter.
Use ORTHO or Polar Tracking to control the angle.
7. Repeat the previous step to create more copies or press the *Space bar* or *Enter* key to stop.

To mirror entities using grips

1. Select the entity or entities.
2. Select the grip or grips you want to manipulate.

3. Click the grip you want to use as the base point for the mirroring.
The *Grip Stretch* command is launched.
4. Do one of the following:
 - Right click, then select *Mirror* in the context menu.
 - Repeatedly hit the *Space bar* or *Enter* key to cycle through the *Grips Editing* commands until *Mirror* is active.

The *GRIP_MIRROR* prompt box displays.



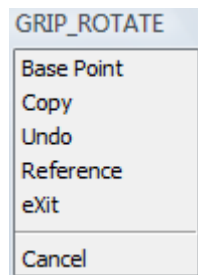
The command bar reads: Enter second point: Base point/Copy/Undo/eXit.

5. Specify a second point to define the mirror axis.
The selection is mirrored.

To rotate entities using grips

1. Select the entity or entities.
2. Select the grip or grips you want to manipulate.
3. Click the grip you want to use as the base point for the rotation.
The *Grip Stretch* command is launched.
4. Do one of the following:
 - Right click, then select *Rotate* in the context menu.
 - Repeatedly hit the *Space bar* or *Enter* key to cycle through the *Grips Editing* commands until *Rotate* is active.

The *GRIP_ROTATE* prompt box displays.



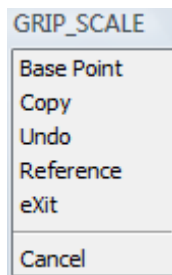
The command bar reads: Enter rotation angle: Base point/Copy/Undo/eXit.

5. Do one of the following to rotate the selection:
 - Specify a point to define the rotation angle.
 - Type the rotation angle in the command bar, then press Enter

To scale entities using grips

1. Select the entity or entities.
2. Select the grip or grips you want to manipulate.
3. Click the grip you want to use as the base point for the scaling.
The *Grip Stretch* command is launched.
4. Do one of the following:
 - Right click, then select *Scale* in the context menu.
 - Repeatedly hit the *Space bar* or *Enter* key to cycle through the *Grips Editing* commands until *Scale* is active.

The *GRIP_SCALE* prompt box displays.



The command bar reads: Enter scale factor: Base point/Copy/Undo/eXit.

5. Type the scale factor in the command bar, then press Enter to scale the selection.

Selecting Entities

Before you can modify one or more entities, they must be selected.

Quick Select allows to select entities using different criteria such as entity type, color, line type or any other entity property.

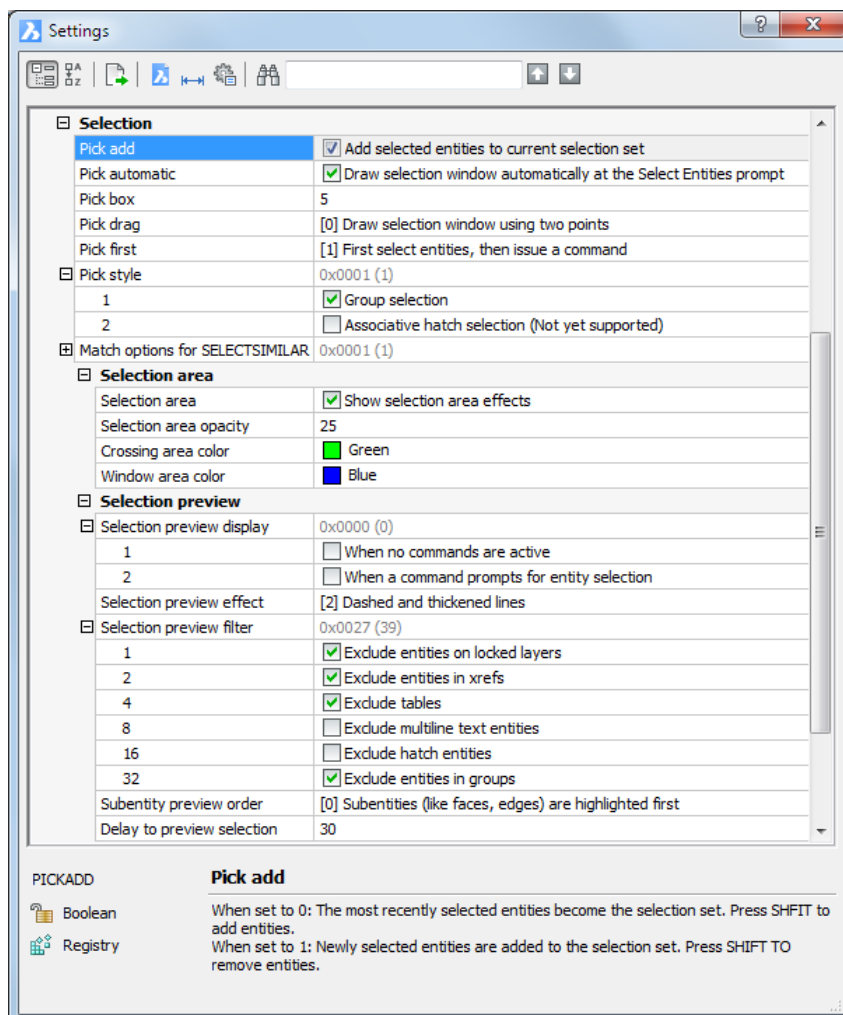
A number of system variables control the selection methods.


Selection and grip settings

When you select entities in a drawing, the selection method and the display of selected entities, e.g. whether grips display or not, is controlled by a number of settings.

Selection settings

Selection settings sit in the *Selection* settings group of the *Program Options* settings category in the Settings dialog.



Name	System Variable	Description
Highlight	HIGHLIGHT	Determines whether or not entities highlight on screen when selected.
Image highlight	IMAGEHLT	Determines whether the entire raster image highlights or only the raster image frame when selected.
Pick add	PICKADD	Controls whether subsequent selections replace the current selection set or add to it. <i>ON</i> : The selection set is extended if you select additional entities. Press and hold the Shift key to remove entities from the selection set. <i>OFF</i> : You cannot add entities to a selection set. The newly selected entity or entities replace the existing selection set. However, if you press and hold the <i>Shift</i> key, you can add entities. If you select an entity that was already selected while pressing the <i>Shift</i> key, this entity is removed from the selection set.
Pick automatic	PICKAUTO	Controls automatic windowing at the Select Objects prompt. <i>ON</i> : The <i>Window-Inside</i> and <i>Crossing Window</i> options of the <i>Select</i> command are chosen by default. This method is referred to as <i>Automatic Windowing</i> . <i>OFF</i> : You need to explicitly specify a selection method.
Pick box	PICKBOX	Defines the size of the small square at the end of the selection cursor (). If you select an entity by clicking the <i>Pick Box</i> must touch or overlap the entity. The default size of the <i>Pick Box</i> is 3.
Pick drag	PICKDRAG	Controls the method of drawing a selection window. <i>ON</i> : Allows to define a selection window by dragging: press and hold the left mouse button to define the first corner of the rectangle, then move the mouse to define the size of the selection window and release the mouse button to define the opposite corner. <i>OFF</i> : Define the selection window by clicking two opposite corners.
Pick first	PICKFIRST	Controls whether you select objects before or after you issue a modification command. <i>ON</i> : Allows to first compose a selection set, then launch a modification command. <i>OFF</i> : You must first start the command, then compose the select entities.
Pick style	PICKSTYLE	Controls the selection of groups and associative hatches. 0 = No group selection, nor associative hatch selection. 1 = Group selection: if a member of a selectable group is selected, all members of the group are selected. 2 = Associative hatch selection: the hatch and




		its boundary are selected, no matter what is picked: the hatch or the boundary (not supported yet). 3 = Group selection and Associative hatch selection.
Match options for SELECTSIMILAR	SELECTSIMILARMODE	Controls which properties must match for an entity of the same type to be selected with the SELECTSIMILAR command.
Selection area	SELECTIONAREA	Controls the display of selection area effects.
Selection area opacity	SELECTIONAREAOPACITY	Controls the transparency of the selection area: the higher the value the more opaque the selection area.
Crossing area color	CROSSINGAREACOLOR	Specifies the color for crossing selection areas.
Window area color	WINDOWAREACOLOR	Specifies the color for window selection areas.
Selection preview display	SELECTIONPREVIEW	Controls when entities highlight when the cursor hovers over them: 0=never; 1=when no commands are active; 2=when a command prompts for entity selection; 3= both, when no commands are active and when a command prompts for entity selection.
Selection preview effect	PREVIEWEFFECT	Controls the appearance of highlighted entities: 0=dashed lines; 1=thickened lines; 2=dashed and thickened lines (not yet supported).
Selection preview filter	PREVIEWFILTER	Controls which entity types are excluded from selection previewing.
GL Selection Highlight Style	HIGHLIGHTEFFECT	Specifies the highlight method: 0=dashed lines; 1=use different color; 2=use thickened line; 3=use different color and thickened line.
Selection Highlight color	HIGHLIGHTCOLOR	Specifies the highlight color to be used when HIGHLIGHTEFFECT is 1 or 3.
Subentity preview order	PREVIEWTOPDOWN	Controls the order in which subentities of an entity are previewed and selected. 0 = Subentities, such as faces and edges, highlight first. 1 = The top level entity, e.g. the whole solid, highlights first. Repeatedly pressing the TAB key highlights all entities one by one.
	User Preference	
Preview selection delay	PreviewDelay	Specifies how many milliseconds to wait before highlighting the (sub)entities under the cursor.

Grips settings

If the *Grips* system variable is *ON*, grips display on selected entities.

Grips settings sit under *Display/Viewing* in the *Drafting* settings group of the *Drawing* settings category in the Settings dialog.

Grips can be used to manipulate entities (see *Grips Editing*)

Grips	
Grips	<input checked="" type="checkbox"/> Turn on grips
Grips in blocks	<input type="checkbox"/> Assigns grips to entities within the block
Grip color	 72
Selected grip color	 240
Hover grip color	 150
Grip size	4
Grip object limit	100
Grips attraction	<input type="checkbox"/> Enable grips attraction
Grips attraction distance	3

Name	System Variable	Description
Grips	GRIPS	Toggles the display of grips on/off.
Grips in blocks	GRIPBLOCK	Toggles the display of grips on entities in blocks on/off.
Grip color	GRIPCOLOR	Sets the color of unselected grips. Default value is green (index color no. 72).
Selected grip color	GRIPHOT	Sets the color of selected grips, so called 'hot grips'. Default value is red (index color no. 240).
Hover grip color	GRIPHOVER	Sets the color of an unselected grip when the cursor pauses over it. Default value is blue (index color no. 150).
Grip size	GRIPSIZE	Sets the size of the grip box, expressed in pixels. Default value = 4.
Grip object limit	GRIPOBJLIMIT	Suppresses the display of grips if the number of selected objects exceeds the value of GRIPOBJLIMIT. If set to 0 (zero) grips always display.
Grips attraction	EnableAttraction (*)	Toggles the magnetism of grips on/off.
Grips attraction distance	AttractionDistance (*)	Sets the grips attraction distance: if the cursor is within this distance the cursor is attracted by the grip and jumps to it.

(*) User Preference

Selection Methods

Before you start to use modification commands, you need to know how to compose a selection set. If the *PICKFIRST* variable is *ON* you can build the selection set before starting the command, else you are prompted to select the entities after you launched the modification command.

Using *QUICK SELECT* you can specify filters, such as entity type, color, linetype or any other entity property, to compose a selection set.

The *SELECTIONPREVIEW* and *PREVIEWEFFECT* system variables facilitate the selection procedure.

Composing a selection set before (pre-pick)

To compose a selection set before launching a modification command do one or more of the following:

- Click an entity.
The entity highlights and grips show.

NOTE If the Quad cursor menu is active you must press and hold the Ctrl key while selecting entities.
Click again to unselect.

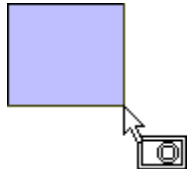
- Click to define the first corner of a selection window.
Move the mouse to the left to define a *Crossing Window*:



All entities which overlap the window or are completely inside the window are added to the selection set.

The selection window displays in dashed lines with a green background.

- Click to define the first corner of a selection window.
Move the mouse to the right to define a *Window-Inside*:



All entities which are completely inside the window are added to the selection set.

The selection window displays in continuous lines with a blue background.

- Press and hold the *Shift* key, then use one of the above methods to select entities that you want to *remove* from the selection set.

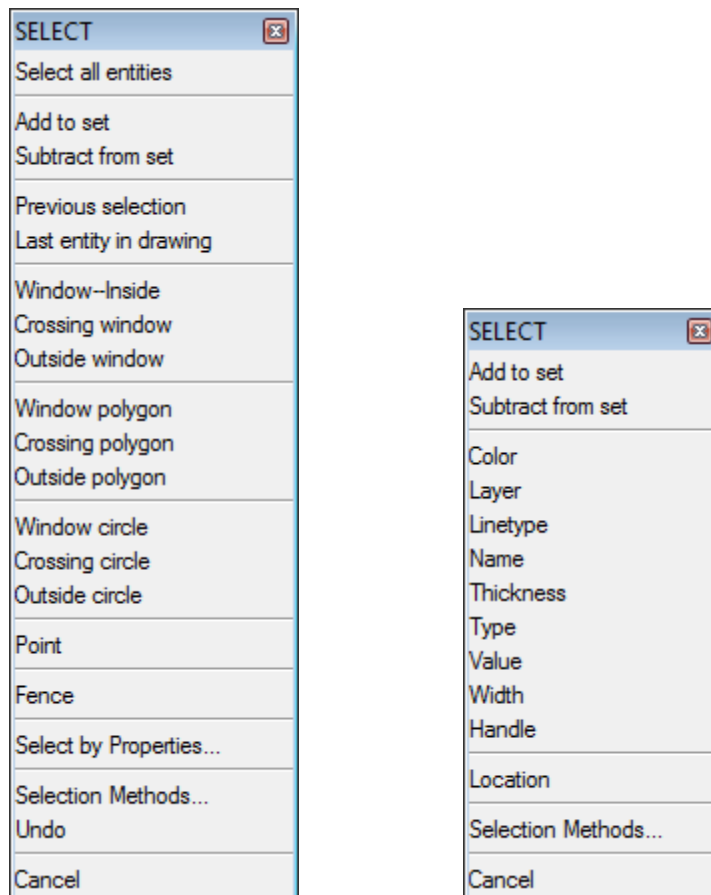
NOTE If the *PICKADD* system variable is *OFF*, you cannot add entities to a selection set.

Composing a selection set afterwards (post-pick)

Command: SELECT

- If you launch a modification command when no selection set is active, you are prompted to select entities. BricsCAD provides a range of selection methods to let you compose your selection set easily.
- The various selection methods are:
 - Add to set*: Selected entities are added to the selection set.
 - Subtract from set*: Selected entities are removed from the selection set.
 - Picking*: Place the pickbox over a part of the object and click.
 - Select all entities*: Select all entities in the entire drawing, including entities which are on hidden layers. Entities on frozen layers are not selected.
 - Previous selection*: Re-use the previous selection set.
 - Last entity in drawing*: Select the most recent entity.
 - Window*: All entities which are inside, overlap or are completely outside a window are selected. You can choose between a *rectangle*, *polygon* or a *circle*.
 - Fence*: All entities that cross a multi-segment line are selected.
 - Point*: Click a point to select all closed entities which enclose this point.

- *Select by Properties*: Displays the *Select by Properties* context menu to select entities by *Color, Layer, Linetype, Name, Thickness, Type, Value, Width* or *Handle*.
- *Undo*: Undoes the last selection action.
- *Location*: Displays the *Select Location* context menu.



Select Location and Select By Properties context menus

Selecting overlapping entities

When entities are close together or lie on top of each other it is difficult or sometimes impossible to select the entities needed.

Cycling through the overlapping entities, highlighting each entity one by one helps you to select such entities.

To select overlapping entities if SELECTIONPREVIEW is ON or OFF:

1. Press and hold the Shift key then press and hold the space bar.
2. Click on the overlapping entities.
The entity lying on top highlights.
The command bar reads: <Cycle on>
3. Release the space bar, then the Shift key.
4. Click anywhere in the drawing to highlight the next entity.
5. Right click, hit the space bar or press Enter to select the highlighted entity.
The highlighted entity is selected.
The command bar reads: <Cycle off>
6. (option) Repeat steps 1 through 5 to select more entities.

To select overlapping entities if SELECTIONPREVIEW is ON:

1. Move the cursor to the overlapping entities.
The entity lying on top highlights.
2. Press and hold the shift key.
3. Hit the space bar repeatedly.
The overlapping entities highlight one by one.
4. Release the shift key, then click to select the currently highlighted entity.
5. (option) Repeat steps 1 through 4 to select more entities.

Quick Select




Using Quick Select (Internet connection needed)

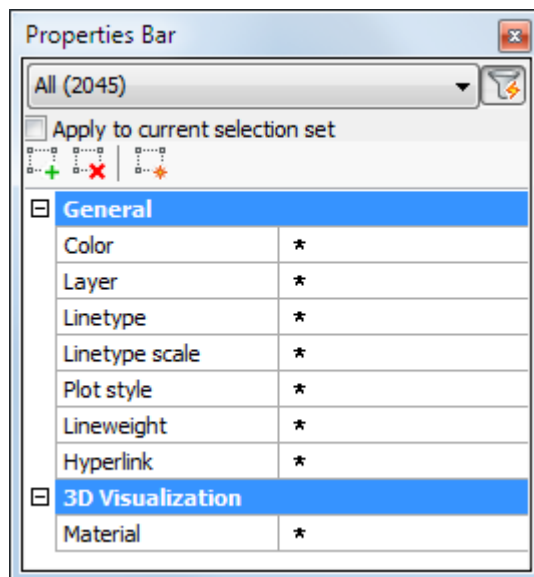
Using *Quick Select* you can specify filters, such as entity type, color, linetype or any other entity property, to compose a selection set.

After composing the filter criteria you can choose to add the matching entities to a new selection set, add the entities to the current selection set or remove them from the current selection.

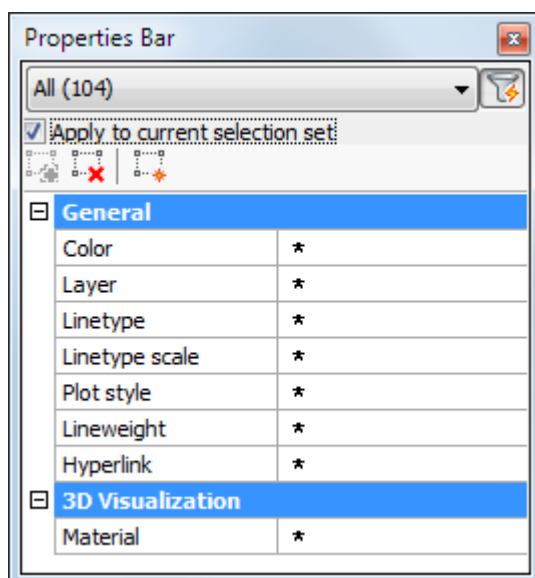
Quick Select applies to the entire drawing or to a previously created selection.

To create a selection set

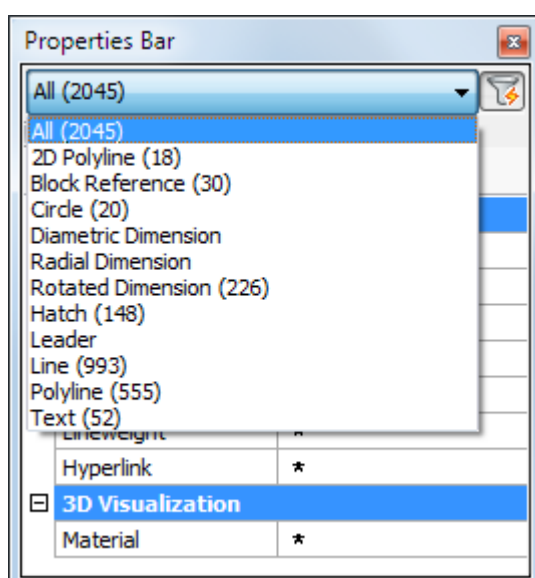
1. Click the *Quick Select* tool button () on the Properties bar.
The *Properties bar* mode is set to *Quick Select*.



2. (option) Click the *Apply to current selection* option if you want to filter an existing selection set.



- Click the *Selection List* button, to see a list of all entity types in the current drawing or current selection set (see previous step).
A list of all entities in the drawing and the number of entities of each type displays.






- Select an entity type in the list, e.g. *Line*.
Line properties show in the *Properties Bar*.

Properties Bar

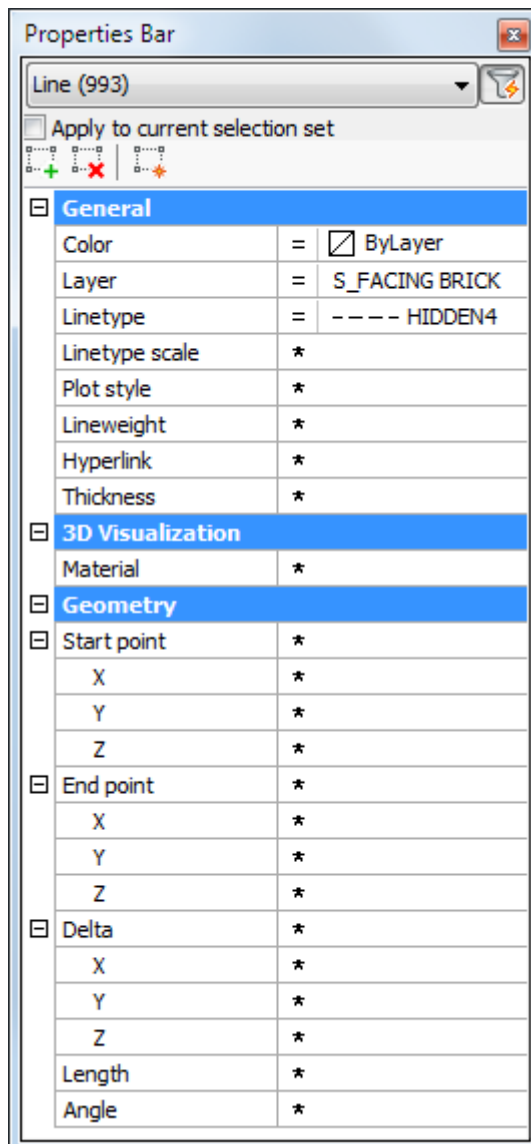
Line (993)

☐ Apply to current selection set

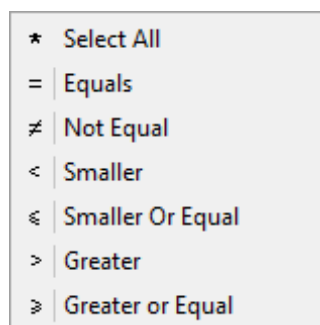
  

General	
Color	*
Layer	*
Linetype	*
Linetype scale	*
Plot style	*
Lineweight	*
Hyperlink	*
Thickness	*
3D Visualization	
Material	*
Geometry	
Start point	*
X	*
Y	*
Z	*
End point	*
X	*
Y	*
Z	*
Delta	*
X	*
Y	*
Z	*
Length	*
Angle	*

5. Specify the property filters, e.g. *Color*, *Layer* and *Linetype*.



6. (option) Specify a different selection operator for each entity property: Click the = sign, then select an operator in the list.





7. Do one of the following:
- Click the *Add to new selection set* button (the orange star icon). The entities that match the selected criteria are selected.

Properties Bar

Line (12)

General	
Color	ByLayer
Layer	S_FACING BRICK
Linetype	--- HIDDEN4
Linetype scale	2.54
Plot style	ByColor
Lineweight	ByLayer
Hyperlink	
Thickness	0.00
3D Visualization	
Material	ByLayer
Geometry	
Start point	*Varies*
X	*Varies*
Y	*Varies*
Z	0.00
End point	*Varies*
X	*Varies*
Y	*Varies*
Z	0.00
Delta	*Varies*
X	*Varies*
Y	*Varies*
Z	0.00
Length	5306.48
Angle	*Varies*

- Click the *Add to the current selection set* button ().
The entities that match the selected criteria are added to the current selection.
 - Click the *Remove from current selection set* button ().
The entities that match the selected criteria are removed from the current selection.
8. (option) Repeat the above procedure to further refine the selection set.

Rearranging Entities

The following tools to modify the location or orientation of existing entities are available:

- Move a selection set
- Rotate a selection set about a specified point
- Change the insertion point, height, style and rotation angle of single line texts.
- Change the insertion point and rotation angle of blocks.
- Modify the display order of overlapping entities.

Moving Entities

Commands: MOVE, CUTCLIP, PASTECLIP, PASTEORIG and PASTEBLOCK

The *Move* command moves entities in the drawing.

The *Cutclip* command copies entities to the *Clipboard*, and then erases them from the drawing.

The *Pasteclip* command pastes entities from the *Clipboard* into the current drawing (short for "paste clipboard").

The *Pasteorig* command pastes entities from the *Clipboard* into another drawing, using the coordinates of the source drawing.

The *Pasteblock* command pastes entities from the *Clipboard* as a block into the current drawing.

Use *Cutclip* then either *Pasteclip*, *Pasteorig* or *Pasteblock* to move entities to another drawing.

The Copy option of the Grips Editing commands allows to create multiple copies of the entities being stretched, moved, rotated, mirrored or scaled.


NOTE When DRAGSNAP is ON, dragged entities display at the current entity snap location.

Moving entities in a drawing

The default method is to create a selection set and then specify a starting point (base point) and an endpoint (displacement point) to define the relocation of the entities. You can also move the entities using a direction vector.

Some entities can be moved using grips. The grip you select depends on the type of entity. For example, to move a line entity, select the midpoint grip. To move a curved entity, such as an arc, circle, or ellipse, select the center point grip. Not all entities can be moved using grips.

Using the Move command

1. Do one of the following:
 - Click the *Move* tool button () on the *Modify* toolbar.
 - Choose *Move* in the *Modify* menu.
 - Type *move* or *M* in the command bar.

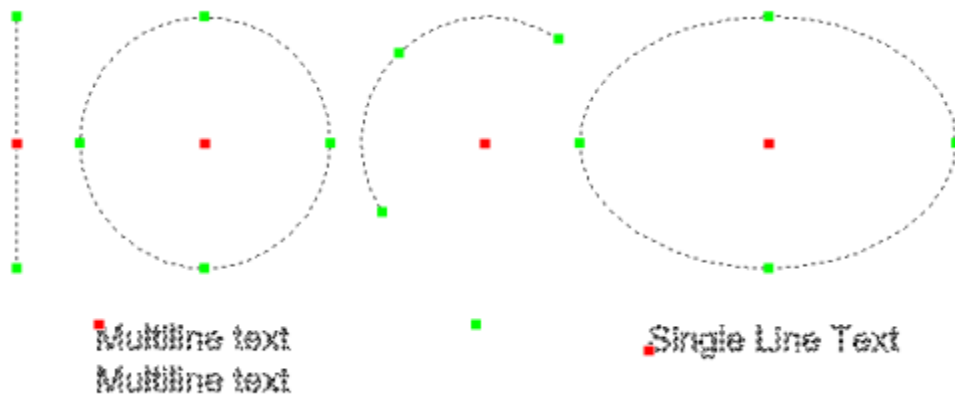
The command bar reads: Select entities to move:
2. Select the entities, then right click or press Enter.
The command bar reads: Vector/<Base point>:
3. Specify the base point.
The selection set is now attached to the cross hairs.
The command bar reads: Displacement point.
4. Do one of the following to specify the displacement point.

- Click the displacement point.
- Use Direct Distance Entry: type the displacement distance, then press Enter.
The distance is measured in the cursor direction.
Use ORTHO or POLAR TRACKING to constrain the movement of the cross hairs.

The selection set is moved.

Moving entities using grips

1. Select the entity.
2. Click the grip.
The selected grip displays in the *Selected Grip Color* (defined by the *GRIPHOT* system variable, default = RED).
The entity moves with the cross hairs.



3. Click to relocate the entity.

NOTE The following entities can be moved using grips:

- lines, infinite lines and rays
- circles and circular arcs
- ellipses and elliptical arcs
- 3D solids
- text and multi-line text
- blocks

Moving entities between drawings

Entities can be moved between drawings in one of three ways:

Use *CUTCLIP* to cut the entities from the source drawing, then either *PASTECLIP*, *PASTEORIG* or *PASTEBLOCK* to paste the entities into the target drawing.

NOTE Use Ctrl + Tab (press and hold the Ctrl key, then press the Tab key) to cycle between open drawings.

Using Cut and Paste to move entities between drawings

1. In the source drawing, select the entities you want to move.

2. Do one of the following:
 - Right click, then choose *Cut* in the context menu.
 - Choose *Cut* in the *Edit* menu.
 - Press *Ctrl* + *X* (press and hold the Ctrl key, then press X).

The entities are deleted in the source drawing and copied to the Clipboard.

3. Switch to the target drawing.

4. Do one of the following:
 - Right click, then choose *Paste* in the context menu.
 - Choose *Paste* in the *Edit* menu.
 - Press *Ctrl* + *V* (press and hold the Ctrl key, then press V).

The bottom left corner of the bounding rectangle of the selection is attached to the cross hairs in the target drawing.

5. Do one of the following:
 - Specify the displacement point.
 - Press Enter or choose *Paste to Original Coordinates* in the *Edit* menu to paste the selection set using the coordinates of the source drawing.

NOTE If you choose *Undo* in the *Edit* menu in the source drawing, the deletion of the selection set is undone.

Move a selection as a block

1. In the source drawing, select the entities you want to move.

2. Do one of the following:
 - Right click, then choose *Cut* in the context menu.
 - Choose *Cut* in the *Edit* menu.
 - Press *Ctrl* + *X* (press and hold the Ctrl key, then press X).

The entities are deleted in the source drawing and copied to the Clipboard.

3. (option) Switch to the target drawing.

4. Choose *Paste as a Block* in the *Edit* menu or press Ctrl+Shift+V.
The bottom left corner of the bounding rectangle of the selection is attached to the cross hairs in the target drawing.

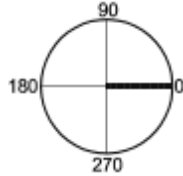
5. Specify the insertion point.

Rotating Entities

Commands: ROTATE and ROTATE3D

You can rotate entities about a specified point at a specified rotation angle or by an angle referenced to a base angle. The default method rotates the entities using a relative rotation angle from their current orientation.

By default, angles start at 3 o'clock and increase in a counter-clockwise direction. If you want to rotate in a clockwise direction you can enter a negative angle by using a minus sign.



NOTES


The *Angular Base* is controlled by the *ANGBASE* system variable, which in turn refers to the current UCS.

The *Angle Direction* is controlled by the *ANGDIR* system variable, which sets the positive angle direction from angle 0 with respect to the current UCS.

The Copy option of the Grips Editing commands allows to create multiple copies of the entities being stretched, moved, rotated, mirrored or scaled.

Rotating a selection set

1. Do one of the following:

- Click the *Rotate* tool button () on the *Modify* toolbar.
- Choose *Rotate* in the *Modify* menu.
- Type *rotate* or *RO* in the command bar.

The command bar reads: Select entities to rotate:

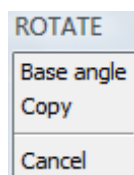
2. Select the entities, then right click or press Enter.

The command bar reads: Rotation point:

3. Specify the rotation point.

The command bar reads: Copy/Base angle/<Rotation angle>:


A prompt menu displays:



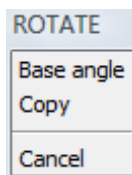
4. (option) Choose *Copy* in the prompt menu or type *C*, then press Enter to create a rotated copy of the selection set.
5. Specify the rotation angle.
The selection set is rotated.

NOTE Use the *Center* option of the Array command if you want to keep the original entities.

Rotating a selection set in reference to a base angle

1. Do one of the following:
 - Click the *Rotate* tool button () on the *Modify* toolbar.
 - Choose *Rotate* in the *Modify* menu.
 - Type *rotate* or *RO* in the command bar.

The command bar reads: Select entities to rotate:
2. Select the entities, then right click or press Enter.
The command bar reads: Rotation point:
3. Specify the rotation point.
The command bar reads: Copy/Base angle/<Rotation angle>:
A prompt menu displays:




4. (option) Choose *Copy* in the prompt menu or type *C*, then press Enter to create a rotated copy of the selection set.
5. Do one of the following:
 - Choose *Base angle* in the prompt menu.
 - Type *B* in the command bar, then press Enter.

The command bar reads: Base angle <0>:
6. To specify the base angle do one of the following:
 - Type the base angle in the command bar.
 - Click the rotation point again, then click a second point.

The command bar reads: New Angle.
7. Do one of the following:
 - Type the new angle in the command bar.
 - Click a point to define the new angle.

Rotating in 3D

1. Do one of the following.
 - Click the *3D Rotate* tool button () on the *Modify* toolbar.
 - Choose *3D Rotate* in the *Modify* menu.
 - Type *rotate3d* in the command bar.

The command bar reads: Select entities to rotate:
2. Select the entities, then right click or press Enter.
The command bar reads: Select axis by: Entity/Last/View/Xaxis/Yaxis/Z-axis/<2 points>:
3. Press Enter to define the rotation axis by specifying two points.
4. Specify the first rotation axis point.
5. Specify the second rotation axis point.
The command bar reads: Reference/<Rotation Angle>:
6. Specify the rotation angle.
The selection set is rotated clockwise, looking in the rotation axis direction.

Mirroring Entities

Commands: MIRROR and MIRROR3D


The *Mirror* command creates a mirror image of a selection set. The selection is mirrored about a mirror line, which you define by specifying two points. You can choose to either delete or retain the original entities.

The *Mirror3d* commands creates a mirror image of selected entities in three-dimensional space. In this case the selection is mirrored about a mirror plan, which can be defined by either specifying three points, selecting an existing two-dimensional planar entity, aligning the plane parallel to the xy, yz, or xz plane of the current UCS or aligning the plane with the current view. You can choose to either delete or retain the original entities.

The Copy option of the Grips Editing commands allows to create multiple copies of the entities being stretched, moved, rotated, mirrored or scaled.


NOTE Whether text is mirrored or not by the *Mirror* command is controlled by the *Mirror Text* (*MIRRTXT*) system variable.

Mirroring entities about a line

1. Do one of the following:
 - Click the *Mirror* tool button () on the *Modify* toolbar.
 - Choose *Mirror* in the *Modify* menu.
 - Type *mirror* or *MI* in the command bar.

The command bar reads: Select entities to mirror:
2. Select the entities then right click or press Enter.
The command bar reads: Start of mirror line.
3. Specify the start point of the mirror line.
4. Specify the endpoint of the mirror line.
The command bar reads: Delete the original entities? <N>
5. Do one of the following:
 - Press Enter to keep the original entities.
 - Type *Y*, then press Enter to delete the original entities.
 - Choose *Yes-Delete entities* in the context menu.

Mirroring entities about a plane

1. Do one of the following:
 - Click the *3D Mirror* tool button () on the *Modify* toolbar.
 - Choose *3D Mirror* in the *Modify* menu.
 - Type *mirror3d* in the command bar.

The command bar reads: Select entities:
2. Select the entities then right click or press Enter.
The command bar reads: Define mirror plane by:
Entity/Last/View/Zaxis/XY/YZ/ZX/<3points>:
3. Specify the first point of the mirror plane.
4. Specify the second point of the mirror plane.
5. Specify the third point of the mirror plane.
The command bar reads: Delete the original entities? <N>

6. Do one of the following:
 - Press Enter to keep the original entities.
 - Type Y, then press Enter to delete the original entities.
 - Choose *Yes-Delete entities* in the context menu.


Aligning Entities

Command: ALIGN

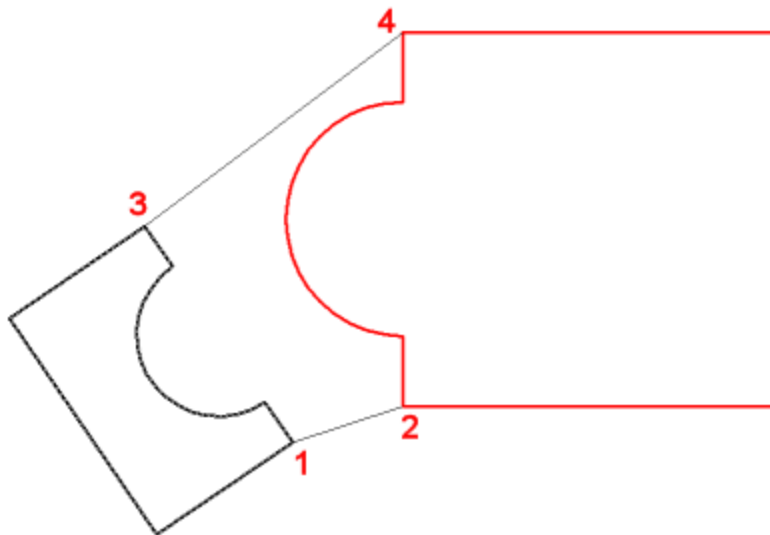
The *Align* command lets you reposition a selection set with respect to an existing entity in the drawing. The selection set is moved and rotated in a single action. If necessary you can even scale the selection to fit in its new location.

The command can be used both in a 2D and 3D environment.

Aligning an entity in 2D

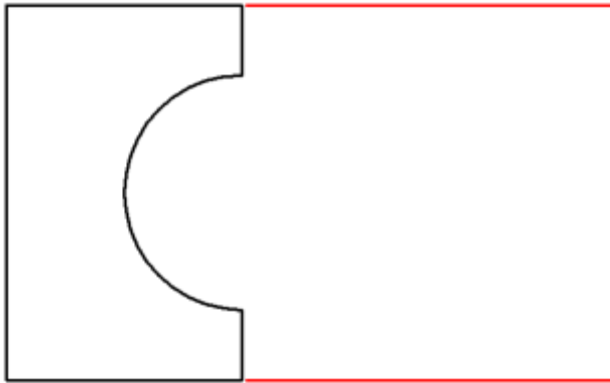
1. Do one of the following:
 - Click the *Align* tool button () on the *Modify* toolbar.
 - Choose *Align* in the *Modify* menu.
 - Type *align* in the command bar, then press Enter.

The command bar reads: Select entities:
2. Select the entities, then right click or press Enter.
The command bar reads: Specify first source point:
3. Snap to the first source point in the selection set (1).
The command bar reads: Specify first destination point:
4. Snap to the first destination point on the reference entity (2).
A witness line is drawn between the source point and the target point.
The command bar reads: Specify the second source point:




5. Snap to the second source point in the selection set (3).
The command bar reads: Specify second destination point:
6. Snap to the second destination point on the reference entity (4).
A witness line is drawn between the source point and the target point.
The command bar reads: Specify the third source point:
7. Right click to skip the third source point.
The command bar reads: Scale objects based on alignment points [Yes/No] <No>:

8. Do one of the following:
 - Press Enter if you don't want to scale the selection set.
 - Type Y and press Enter to scale the selection set.



Result of the Align procedure with scaled selection set.

Aligning an entity in 3D

1. Do one of the following:
 - Click the *Align* tool button () on the *Modify* toolbar.
 - Choose *Align* in the *Modify* menu.
 - Type *align* in the command bar, then press Enter.

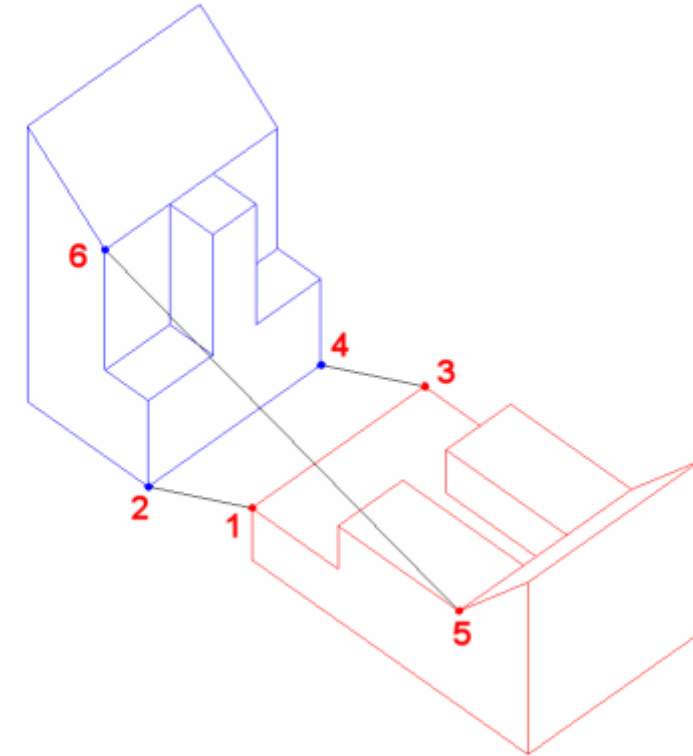
The command bar reads: Select entities:
2. Select the entities, then right click or press Enter.

The command bar reads: Specify first source point:
3. Snap to the first source point in the selection set (1).

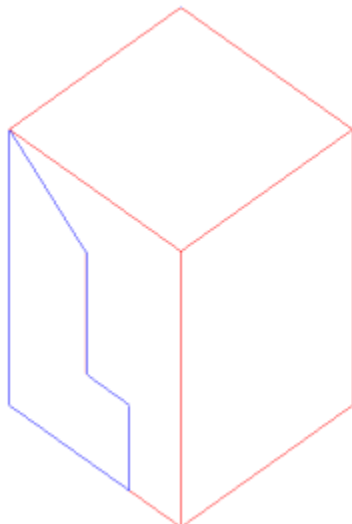
The command bar reads: Specify first destination point:
4. Snap to the first destination point on the reference entity (2).

A witness line is drawn between the source point and the target point.

The command bar reads: Specify the second source point:



5. Snap to the second source point in the selection set (3).
The command bar reads: Specify second destination point:
6. Snap to the second destination point on the reference entity (4).
A witness line is drawn between the source point and the target point.
The command bar reads: Specify the third source point:
7. Snap to the third source point in the selection set (5).
The command bar reads: Specify third destination point:
8. Snap to the third destination point on the reference entity (6).
A witness line is drawn between the source point and the target point.
The entity is moved.



Changing Entities

Command: CHANGE

The *Change* command lets you change the insertion point and orientation of texts and block inserts.

The *Properties* option of the command can also change the layer, line type, line type scale, line weight, etc. but the BricsCAD Properties bar is much more versatile to do such things.

To change text entities

1. Type *change* in the command bar, then press Enter.
The command bar reads: Select entities to change:
2. Select the text entities, then right click or press Enter.
The command bar reads: Change: Entities/Properties/<Change point>:
3. Right click or press Enter to accept the default option.
The command bar reads: New point for text, or Enter for no change:
The first text entity is attached to the cursor.
4. Specify the new insertion point for the text or right click to accept the current location.
The command bar reads: New text style <current style>:
5. Type a new text style and press Enter or right click to keep the current style.
The command bar reads: New height <current height>:
6. Type a new height and press Enter or right click to keep the current height.
The command bar reads: New rotation angle <current angle>:
7. Type a new angle and press Enter or right click to keep the current angle.
The command bar reads: New text <current text>:
8. Type the new text and press Enter or right click to keep the current text.
9. (option) If multiple text entities were selected in step 2, steps 3 through 8 are repeated for each text.

NOTE The *Change* command cannot change Multi-Line text (Mtext) entities.

To relocate blocks

1. Type *change* in the command bar, then press Enter.
The command bar reads: Select entities to change:
2. Select the blocks, then right click or press Enter.
The command bar reads: Change: Entities/Properties/<Change point>:
3. Right click or press Enter to accept the default option.
The command bar reads: New point for block, or Enter for no change:
The first block is attached to the cursor.
4. Specify the new insertion point or right click to accept the current orientation.
The command bar reads: New rotation angle <current angle>:
5. Do one of the following:
 - Right click to accept the current rotation angle.
 - Type a new rotation angle in the command bar, then press Enter.
 - Click to specify the new rotation angle graphically.
6. (option) If multiple blocs were selected in step 2, steps 3 through 5 are repeated for each block.
- 7.

Copying Entities

You can copy a selection set, making one copy or multiple copies within the current drawing or you can also copy entities between drawings.

The following procedures can be used to make a copy of existing entities:

- Create a copy of a selection set within the current drawing
- Create multiple copies of a selection set within the current drawing
- Copy entities between drawings
- Create a copy of a linear entity, aligned parallel to the original
- Create a copy as a mirror image of the original
- Create several copies in a rectangular or circular pattern in 2D
- Create several copies in a rectangular or circular pattern in 3D

NOTE When DRAGSNAP is ON, dragged entities display at the current entity snap location.

Copying Entities in a Drawing

Commands: COPY, COPYCLIP, COPYBASE, PASTECLIP and PASTEBLOCK

You can duplicate entities within the current drawing using the *Copy* command. The default method is to create a selection set and then specify a starting point (base point) and an endpoint (second point) for the copy. You can also specify the displacement as a direction vector (X,Y,Z).


Depending on the *COPYMODE* system variable you can create multiple copies of the selection set or just one.

Copybase and *Pasteblock* pastes the selection as a block.

The Copy option of the Grips Editing commands allows to create multiple copies of the entities being stretched, moved, rotated, mirrored or scaled.

To copy a selection set

1. Do one of the following:

- Click the *Copy* tool button () on the *Modify* toolbar.
- Choose *Copy* in the *Modify* menu.
- Type *copy* or *CO* in the command bar.

The command bar reads: Select entities to copy:

2. Select the entities, then right click or press Enter.

If *Copy Mode* is *Multiple*:

The command bar reads: Enter base point: mOde/<Displacement>:

If *Copy Mode* is *Single*:

The command bar reads: Enter base point: mOde/Multiple/<Displacement>:

3. Specify the base point.

The selection set is now attached to the cross hairs.

The command bar reads: Enter second point: <Enter to use the base point as displacement>:

4. Do one of the following:

- Click the second point.
- Use Direct Distance Entry: type the displacement distance, then press Enter.
The distance is measured in the cursor direction.
Use ORTHO or POLAR TRACKING to constrain the movement of the cross hairs.

- Press <Enter>: the first point is interpreted as a relative X,Y,Z displacement. E.g. if the coordinates of the base point are 2,3,0, the selection set is copied 2 units in the X direction and 3 units in the Y direction from their current position.


The selection set is copied.

If *Copy Mode* is *Single*, the copy command is completed.

If *Copy Mode* is *Multiple*, you are prompted to create the next copy. See To make multiple copies.

To make multiple copies

1. Do one of the following:

- Click the *Copy* tool button () on the *Modify* toolbar.
- Choose *Copy* in the *Modify* menu.
- Type *copy* or *CO* in the command bar.

The command bar reads: Select entities to copy:

2. Select the entities, then right click or press Enter.

If *Copy Mode* is *Multiple*:

The command bar reads: Enter base point: mOde/<Displacement>:

If *Copy Mode* is *Single*:

The command bar reads: Enter base point: mOde/Multiple/<Displacement>:

3. (Option) If *Copy Mode* is *Single*, do one of the following:

- Choose *Multiple* in the prompt menu
- Type *M*, then press Enter

The command bar reads: Enter base point: mOde/<Displacement>:

4. Specify the base point.

The selection set is now attached to the cross hairs.

The command bar reads: Enter second point: <Enter to use the base point as displacement>:

5. Do one of the following to specify the second point.

- Click the second point.
- Type the displacement distance, then press Enter.
The distance is measured in the cursor direction.
Use ORTHO or POLAR TRACKING to constrain the movement of the cross hairs.

The first copy of the selection set is created.

The command bar reads: Enter second point: Undo/Exit:

6. Do one of the following:

- Repeat step 5 to create additional copies.
- Right click, choose *Exit* in the prompt menu or press Enter to stop.
- Choose *Undo* in the prompt menu or type *U* in the command bar, then press enter to delete the previous copy of the selection set.

To paste a selection set as a block

1. Do one of the following:

- Choose *Copy with Base Point* in the *Edit* menu.
- Type *copybase* in the command bar, then press Enter.
- Press and hold the *Ctrl* and *Shift* keys, then press *C*.

The command bar reads: Select base point:

2. Specify the base point (origin point).

The command bar reads: Select entities to copy to the clipboard:

3. Select the entities.
4. Press Enter to conclude the selection of entities.
5. Do one of the following:
 - Choose *Paste as Block* in the *Edit* menu.
 - Type *pasteblock* in the command bar, then press Enter.
 - Press and hold the *Ctrl* and *Shift* keys, then press *V*.

The selection set is attached to the cursor.

The command window reads: Select insertion point.

6. Specify the insertion point.
The selection is pasted as a block.
7. (option) Repeat steps 5 and 6 to paste another copy as a block.

NOTE If you paste multiple copies of the same selection set, each block is named differently.

Copying Entities Between Drawings

Commands: COPYCLIP, COPYBASE, PASTECLIP, PASTEORIG and PASTEBLOCK

Entities can be copied between drawings in several ways:

- *Copyclip* and *Pasteclip* pastes the selection using a default base point.
- *Copyclip* and *Pasteorig* pastes the selection using the coordinates of the source drawing.
- *Copybase* and *Pasteclip* pastes the selection using a user-defined base point.
- *Copybase* and *Pasteblock*: pastes the selection as a block in the target drawing.

NOTE Use Ctrl + Tab (press and hold the Ctrl key, then press the Tab key) to cycle between open drawings.



Copy Between Drawings (Internet connection needed)

Using Copy and Paste to copy entities between drawings

1. In the source drawing, select the entities you want to Copy.
2. Do one of the following:
 - Right click, then choose *Copy* in the context menu.
 - Choose *Copy* in the *Edit* menu.
 - Press *Ctrl* + *C* (press and hold the Ctrl key, then press C).

The entities are copied to the Clipboard.

3. Switch to the target drawing.
4. Do one of the following:
 - Right click, then choose *Paste* in the context menu.
 - Choose *Paste* in the *Edit* menu.
 - Press *Ctrl* + *V* (press and hold the Ctrl key, then press V).

The bottom left corner of the bounding rectangle of the selection is attached to the cross hairs in the target drawing.

5. Do one of the following:
 - Specify the displacement point.
 - Press Enter to paste the selection set using the coordinates of the source drawing.

Using Copy and Pasteorig to copy entities between drawings

1. In the source drawing, select the entities you want to Copy.
2. Do one of the following:
 - Right click, then choose *Copy* in the context menu.
 - Choose *Copy* in the *Edit* menu.
 - Press *Ctrl + C* (press and hold the Ctrl key, then press C).

The entities are copied to the Clipboard.
3. Switch to the target drawing.
4. Do one of the following:
 - Choose *Paste to Original Coordinates* in the Edit menu.
 - Type *pasteorig* in the command bar, then press Enter.

The selection is pasted using the coordinates of the source drawing.

Using Copybase and Paste to copy entities between drawings

1. In the source drawing do one of the following:
 - Choose *Copy with Base Point* in the *Edit* menu.
 - Type *copybase* in the command bar, then press Enter.
 - Press and hold the *Ctrl* en *Shift* keys, then press *C*.

The command bar reads: Select base point:
2. Specify the base point (origin point).
The command bar reads: Select entities to copy to the clipboard:
3. Select the entities.
4. Press Enter to conclude the selection of entities.
5. Switch to the target drawing.
6. Do one of the following:
 - Choose *Paste* in the *Edit* menu.
 - Press *Ctrl + V* (press and hold the Ctrl key, then press V).

The selection set is attached to the cursor.

The command bar reads: Select insertion point:
7. Specify the insertion point to paste the selection set.
8. (option) Repeat step 7 to create more copies of the selection set.

Paste a selection as a block in the target drawing

1. In the source drawing do one of the following:
 - Choose *Copy with Base Point* in the *Edit* menu.
 - Type *copybase* in the command bar, then press Enter.
 - Press and hold the *Ctrl* en *Shift* keys, then press *C*.

The command bar reads: Select base point:

2. Specify the base point (origin point).
The command bar reads: Select entities to copy to the clipboard:
3. Select the entities.
4. Press Enter to conclude the selection of entities.
5. Switch to the target drawing.
6. Do one of the following:
 - Choose *Paste as Block* in the *Edit* menu.
 - Type *pasteblock* in the command bar, then press Enter.
 - Press and hold the *Ctrl* and *Shift* keys, then press *V*.

The selection set is attached to the cursor.

The command window reads: Select insertion point.
7. Specify the insertion point.
The selection is pasted as a block.
8. (option) Repeat steps 5 and 6 to paste another copy as a block.

NOTE If you paste multiple copies of the same selection set, each block is named differently.

Making Parallel Copies


Command: OFFSET

The *Offset* command creates a copy of linear entities and align them parallel to the original entities at a specified distance. You can make parallel copies of arcs, circles, ellipses, elliptical arcs, lines, two-dimensional polylines, rays and infinite lines.

Making parallel copies of curved entities creates larger or smaller curves, depending on which side of the original entity you place the copy. For example, placing a parallel copy of a circle outside the circle creates a larger concentric circle; positioning the copy inside the circle creates a smaller concentric circle.

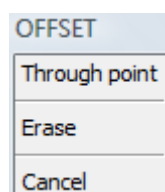
NOTE The *OFFSETGAPTYPE* settings variable controls how potential gaps between segments are treated when closed polylines are offset.

Making a parallel copy at a specified distance

1. Do one of the following:
 - Click the *Offset* tool button () on the *Modify* toolbar.
 - Choose *Offset* in the *Modify* menu.
 - Type *offset* in the command bar.


The command bar reads: Offset: Through point/Erase/<Distance> <current distance>:

A prompt menu displays:



2. (Option) Choose *Erase* in the prompt menu or type *E*, then press Enter. Respond *Yes* at the command prompt to erase the source entity.
3. Do one of the following:
 - Press Enter to accept the current distance.
 - Type a new distance in the command bar.
 - Define a new distance by specifying two points.
 The command bar reads: Select Entity.
4. Select the entity.
The command bar reads: Both sides/<Side for parallel copy>:
5. Click the side for the parallel copy.
The parallel copy is created.
The command bar reads: Select Entity.
6. Do one of the following:
 - Repeat steps 4 and 5 to create more parallel copies.
 - Right click or press Enter to stop.

Making a parallel copy through a point


1. Do one of the following:
 - Click the *Offset* tool button () on the *Modify* toolbar.
 - Choose *Offset* in the *Modify* menu.
 - Type *offset* in the command bar.
 The command bar reads: Offset: Through point/<Distance> <current distance>:
2. Choose *Through point* in the context menu or type *T*, then press Enter.
The command bar reads: Select entity:
3. Select the entity.
The command bar reads: Through point:
4. Specify the point to create the parallel copy through.
The parallel copy is created.
The command bar reads: Select entity:
5. Do one of the following.
 - Repeat steps 3 and 4 to create more parallel copies.
 - Right click or press Enter to stop.

Arraying Entities

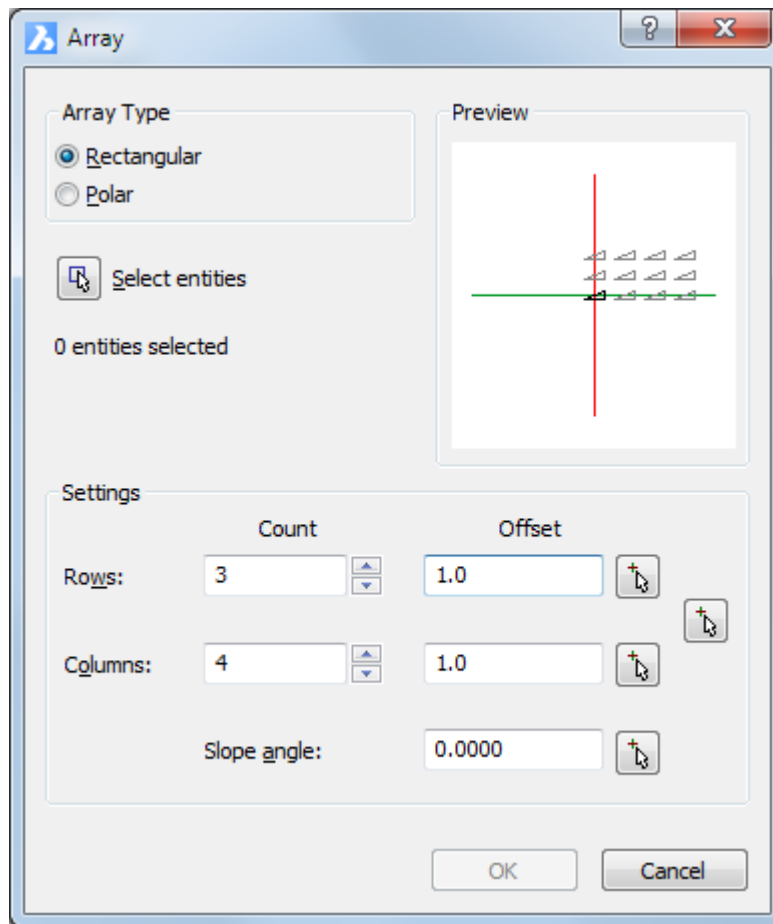
Commands: ARRAY and -ARRAY



The *Array* command copies a selection set in a rectangular or polar (circular) pattern. For a rectangular array you must specify the number of rows and columns and also the spacing between subsequent rows and columns. To create a polar array you will be prompted to specify the center point of the array, the rotation step and the number of items in the array or the angle to fill. You can choose to rotate the selection set about the center point or to keep its original orientation.

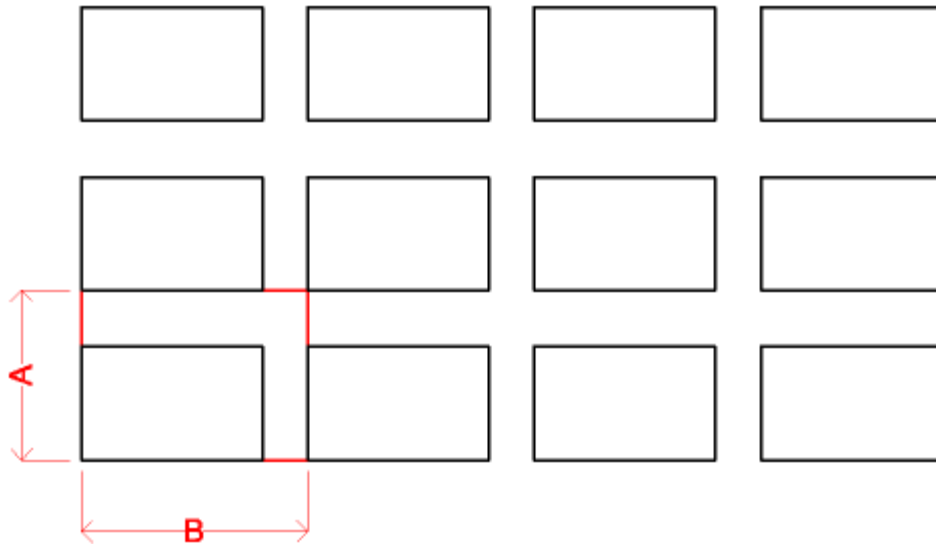
Creating a rectangular array

1. Do one of the following:
 - Click the *2D Array* tool button () on the *Modify* toolbar.
 - Choose *2D Array* in the *Modify* menu.
 - Type *AR* or *array* in the command bar, then press Enter.

A dialog box displays:



2. Under *Array Type* choose *Rectangular*.
3. Click the *Select Entities* button ().
The *Array* dialog box closes temporarily to let you select entities.
4. Select the entities, then right click or press Enter.
The *Array* dialog box reopens.
5. Fill out the *Settings* fields or press the *Pick Points* buttons () to define the settings graphically.
The *Array* dialog box closes temporarily to let you pick point in the drawing.




Row offset (A) and column offset (B) can be defined by the spacing rectangle (red).

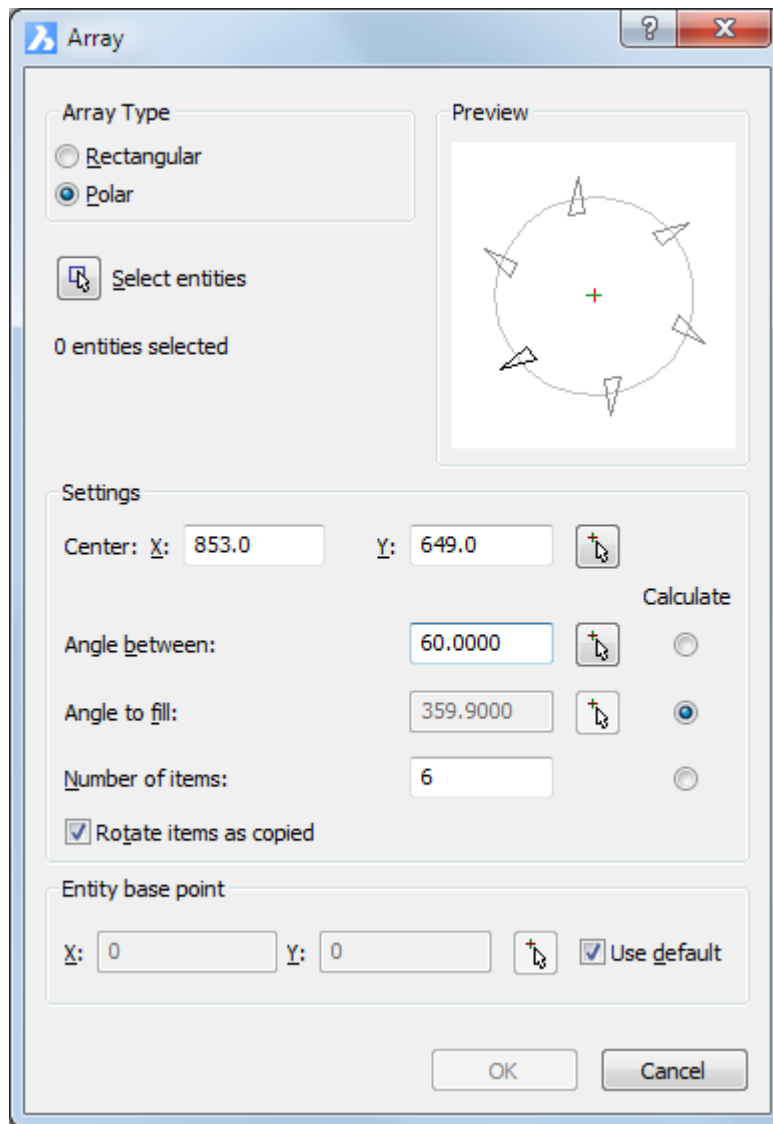
6. Click the OK button to create the array.



NOTE Positive values in steps 4 and 5 are measured along the positive X- and Y-axis of the current UCS. Negative values are measured in the opposite direction.

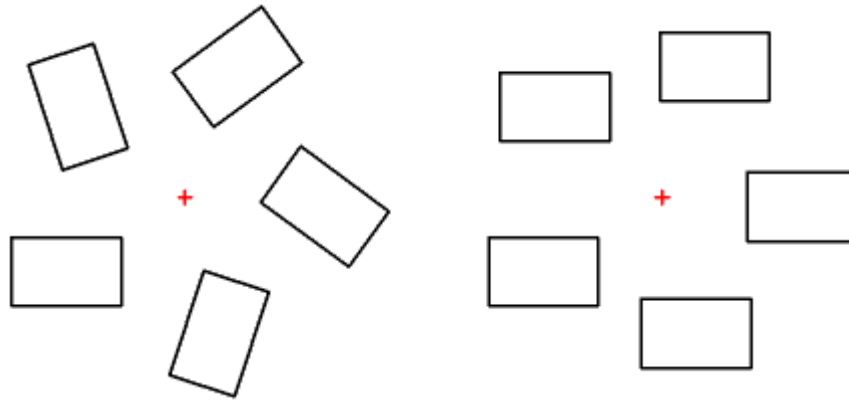
Creating a polar array

1. Do one of the following:
 - Click the *2D Array* tool button () on the *Modify* toolbar.
 - Choose *2D Array* in the *Modify* menu.
 - Type *AR* or *array* in the command bar, then press Enter.

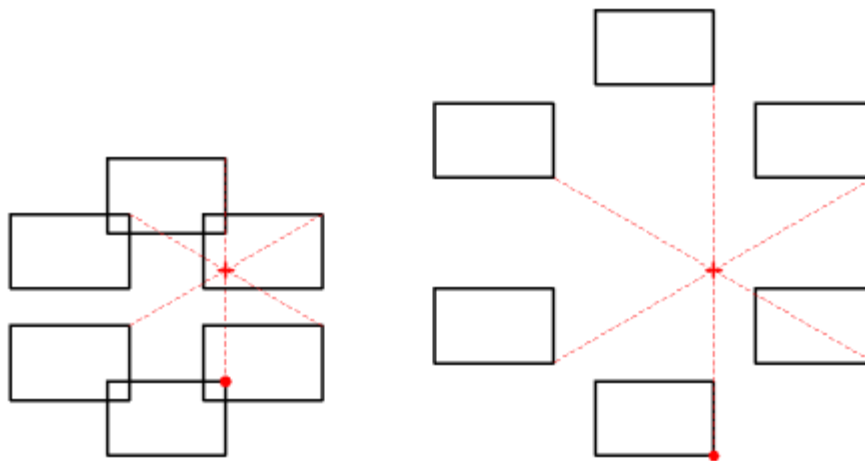
A dialog box displays:



2. Under *Array Type* choose *Polar*.
3. Click the *Select Entities* button ().
The *Array* dialog box closes temporarily to let you select entities.
4. Select the entities, then right click or press Enter.
The *Array* dialog box reopens.
5. Choose a *Calculate* option: *Angle between*, *Angle to fill* or *Number of items*.
The settings field of the selected option is dimmed.
6. Fill out the *Settings* fields or press the *Pick Points* buttons () to define the settings graphically.
The *Array* dialog box closes temporarily to let you pick point in the drawing.
7. (option) Check the *Rotate items as copied* option.



Entities rotated (left) or not (right) about the center point of the array.
8. (option) Define an *Entity base point*.




Polar array with base point (dot) and center point (cross).

Arraying Entities in 3D

Command: 3DARRAY

The *3DArray* command creates multiple copies a selection set in three dimensions. Entities are arrayed in a three-dimensional rectangular (rows, columns, and levels) pattern or a two-dimensional polar (circular) pattern. The polar pattern is created by copying entities about a specified axis. You can choose to rotate the selection set about rotation axis or or to keep its original orientation.

Creating a 3D rectangular array

1. Do one of the following:
 - Click the *3D Array* tool button () on the *Modify* toolbar.
 - Choose *3D Array* in the *Modify* menu.
 - Type *3darray* in the command bar, then press Enter.

The command bar reads: Select entities to array:
2. Select the entities, then right click or press Enter.
The command bar reads: Type of array: Polar/<Rectangular>:

3. Choose *Rectangular* in the prompt menu or type *R* in the command bar and press Enter.

The command bar reads: Number of rows in the array <1>:

4. Type the number of rows in the command bar, then press Enter.
The command bar reads: Number of columns <1>:
5. Type the number of columns in the command bar, then press Enter.
The command bar reads: Number of levels <1>:
6. Type the number of levels in the command bar, then press Enter.
The command bar reads: Vertical distance between rows:
7. To define the distance between rows of the array, do one of the following:
 - Type the distance in the command bar and press Enter.
 - Specify the distance by clicking two points.

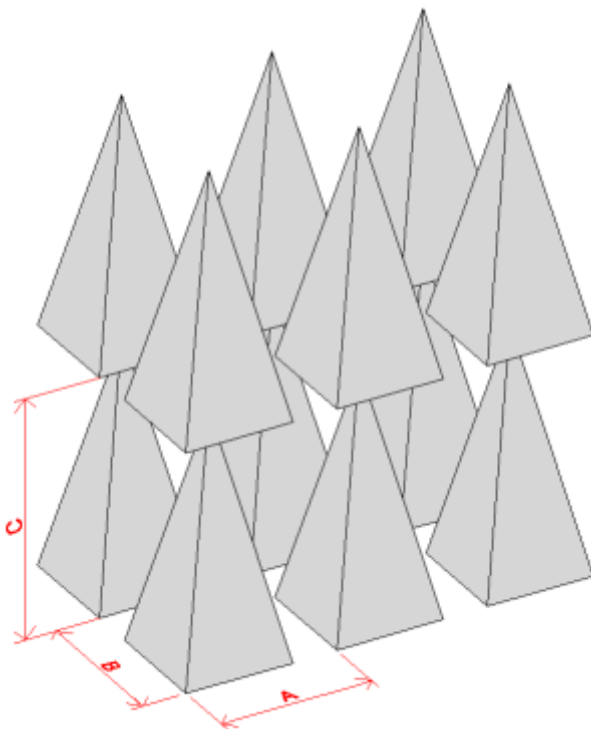
The command bar reads: Horizontal distance between columns:

8. To define the distance between the columns of the array, do one of the following:
 - Type the distance in the command bar and press Enter.
 - Specify the distance by clicking two points.

The command bar reads: Depth between levels:

9. To define the distance between the levels of the array, do one of the following:
 - Type the distance in the command bar and press Enter.
 - Specify the distance by clicking two points.


The array is created.



Rectangular 3D Array: row spacing A, column spacing B, level spacing C

NOTE Positive values in steps 7, 8 and 9 are measured along the positive X-, Y- and Z-axis of the current UCS. Negative values are measured in the opposite direction.

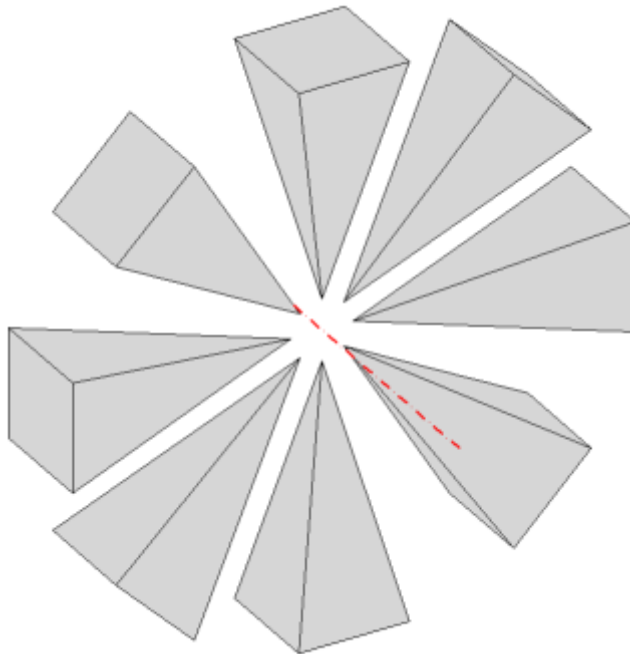
Creating a 3D polar array

1. Do one of the following:
 - Click the *3D Array* tool button () on the *Modify* toolbar.
 - Choose *3D Array* in the *Modify* menu.
 - Type *3darray* in the command bar, then press Enter.

The command bar reads: Select entities to array:
2. Select the entities, then right click or press Enter.
The command bar reads: Type of array: Polar/<Rectangular>:
3. Choose *Polar* in the prompt menu or type *P* in the command bar and press Enter.
The command bar reads: ENTER to specify angle between items/<Number of items to array>:
4. Type the number of items you want in the array, then press Enter.
The command bar reads: Angle to array (+ for ccw, - for cw) <360>:
5. Do one of the following:
 - Type the angle to array, then press Enter.
 - Press Enter to create a 360° array.

The command bar reads: Rotate entities around the array? No/<Yes>
6. Do one of the following:
 - Press Enter to rotate the entities about the array axis.
 - Type N and press Enter to keep the original orientation of the entities.

The command bar reads: Center of polar array.
7. Specify the first point of the array axis.
The command bar reads: Specify second point along central axis of array:
8. Specify the second point of the array axis.
The array is created.



Polar 3D array about a horizontal axis (red).

Resizing Entities

You can resize entities using the following methods:

- Extend entities to a boundary entity
- Trim entities by a cutting entity
- Edit the length of lines, two-dimensional polylines, rays or arcs (length or included angle)
- Stretch
- Scale

Extending Entities

Command: EXTEND and TRIM

The *Extend* command lets you extend entities to a boundary, which is defined by one or more other entities.

If the *Edge Mode* setting (*EDGEMODE*) is *On*, you can extend entities to an implied edge of the boundary entities.

If the boundary entity is not in the same plane as the entity you want to extend, the *Projection Mode* setting (*PROJMODE*) lets you choose how the intersection is to be calculated. The options are:

- Project to the XY plane of the current UCS
- Project to the current view plane
- True 3D mode (No projection).


When extending entities, first select the boundary edges, and then specify the entities to extend, selecting them either one by one or using the fence selection method.

The following entities can be extended: arcs, lines, two-dimensional polylines, rays.

Boundary entities can be: arcs, circles, ellipses, lines, splines, polylines, rays, infinite lines, layout viewports.

To extend entities

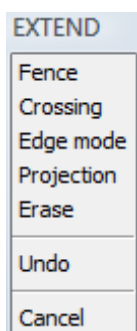
1. Do one of the following:

- Click the *Extend* tool button () on the *Extend/Stretch* flyout of *Modify* toolbar.
- Choose *Extend* in the *Modify* menu.
- Type *extend* or *EX* in the command bar, then press Enter.

The command bar reads: Select boundary entities for extend <ENTER to select all>:

2. Select the boundary entities, then press Enter or right click.

The command bar reads: Fence/Crossing/Edge mode/Projection/eRase/Undo/<Select entity to extend or shift-select to trim>:
A prompt menu displays.



3. Click the entity you want to extend near the end that can make the extension.
The entity is extended.
or
Press and hold the shift key to trim the entity.
4. Repeat step 3 to extend more entities.
5. Right click or press Enter to conclude the *Extend* command.

NOTES Entities which are selected when you launch the *Extend* command will be used as boundary entities.

If you select an entity near the end that cannot make an extension to one of the boundary entities, the *Extend* command is aborted.

Trimming Entities

Command: TRIM and EXTEND

The Trim command lets you clip or trim entities by cutting entities.

If the *Edge Mode* (*EDGEMODE*) system variable is *On*, you can trim entities by an implied edge of the cutting entities.

If the cutting entity is not in the same plane as the entity you want to trim, the *Projection Mode* (*PROJMODE*) system variable defines how the intersection is to be calculated. The options are:

- Project to the XY plane of the current UCS
- Project to the current view plane
- True 3D mode (No projection).


When trimming entities, you first select the cutting edges, and then specify the entities to trim, selecting them either one by one or using the fence selection method.

The following entities can be trimmed: lines, two- and three dimensional polylines, arcs, circles, ellipses, elliptical arcs, splines, rays and infinite lines.

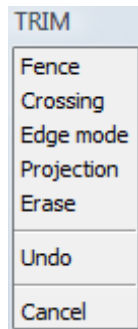
Cutting entities can be: lines, splines, polylines, arcs, circles, elliptical arcs, ellipses, rays, infinite lines, layout viewports.

Entities in a block can be selected as cutting entities.

To trim entities

1. Do one of the following:
 - Click the *Trim* tool button () on the *Modify* toolbar.
 - Choose *Trim* in the *Modify* menu.
 - Type *trim* or *TR* in the command bar, then press Enter.

The command bar reads: Select cutting entities for trim <ENTER to select all>:
2. Select the cutting entities, then press Enter or right click.
The command bar reads: Fence/Crossing/Edge mode/Projection/eRase/Undo<Select entity to trim or shift-select to extend>:
A prompt menu displays.



3. Click the entity you want to trim, the part of the entity that you click will be removed.
The entity is trimmed.
or
Press and hold the shift key to extend the entity.
4. Repeat step 3 to trim more entities.
5. Right click to conclude the *Trim* command.


NOTES Entities which are selected when you launch the *Trim* command will be used as cutting entities.

Changing the length of an entity

Command: LENGTHEN

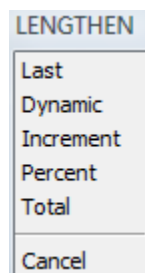
The *Lengthen* command lets you change the length of lines, open polylines and arcs. You can also modify the included angle of arcs.

Change the length of an entity dynamically

1. Do one of the following:
 - Click the *Lengthen* tool button () on the *Modify* toolbar.
 - Choose *Lengthen* in the *Modify* menu.
 - Type *lengthen* in the command bar, then press Enter.

The command bar reads: Edit length: DYnamic/Increment/Percent/Total/<Select entity to list length>:

A prompt menu displays:




2. (option) Click an entity.
The current length of the selected entity displays in the command bar.
In case an arc is selected, the current length and the included angle display in the command bar.

3. Do one of the following:
 - Choose *Dynamic* in the prompt box.
 - Type *DY* in the command bar, then press Enter.

The command bar reads: Mode/<Select entity to change>:
4. Click the entity near the end you want to change.
The length of the entity changes dynamically.
5. Click to define the new length.

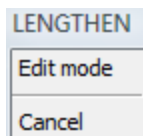
Modify the included angle of an arc

1. Do one of the following:
 - Click the *Edit Length* tool button () on the *Modify* toolbar.
 - Choose *Edit Length* in the *Modify* menu.
 - Type *editlen* in the command bar, then press Enter.

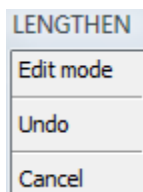
The command bar reads: Edit length: DYnamic/Increment/Percent/Total/<Select entity to list length>:
2. Choose *Total* in the prompt menu or type *T* and press Enter.
The command bar reads: Angle/<Enter total Length (0.00)>:
A prompt menu displays:



3. Choose *Angle* in the prompt menu or type *A* and press Enter.
The command bar reads: Enter total angle <00° 0' 0">:
4. Type the new angle in the command bar and press Enter.
The command bar reads: Mode/<Select entity to change>:
A prompt menu displays:



5. Click the arc at the end you want to lengthen or shorten.
The arc is modified.
The command bar reads: Mode/Undo/<Select entity to change>:
A prompt menu displays:



6. Do one of the following:
 - Select another arc.
 - Choose *Undo* in the prompt menu or type *U* and press Enter to undo the previous action.
 - Choose *Edit mode* in the prompt menu or type *M* and press Enter to choose a different *Edit Length* command option.
 - Choose *Cancel* in the prompt menu to conclude the *Edit Length* command.

NOTE In the Properties bar you can edit the *start angle*, *end angle* and *radius* of an arc.

Stretching Entities


Command: STRETCH

The size and shape of entities can be changed by stretching them. You select an area in your drawing using either a rectangular window or a polygon, then you specify a base point and a displacement point. All points and nodes inside the selected area will be moved over the specified distance. As a result, entities that cross the window or polygon boundary are stretched; those completely within the window or polygon are simply moved.

The Copy option of the Grips Editing commands allows to create multiple copies of the entities being stretched, moved, rotated, mirrored or scaled.

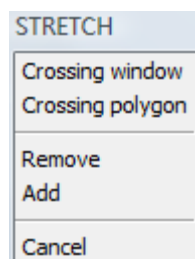
To stretch entities

1. Do one of the following:

- Click the *Stretch* tool button () on the *Extend/Stretch* flyout of the *Modify* toolbar.
- Choose *Stretch* in the *Modify* menu.
- Type *stretch* or *S* in the command bar and press Enter.

The command bar reads: Select entities to stretch by crossing-window or crossing-polygon:

A prompt menu displays:



2. (option) Choose *Crossing window* in the *Stretch* prompt menu, then define the stretch area by a rectangular window.
 3. (option) Choose *Crossing polygon* in the *Stretch* prompt menu, then define the stretch area by a polygon.
 4. (option) Repeat steps 2 and 3 to expand the stretch area.
 5. (option) Choose *Remove* in the *Stretch* prompt menu to select entities in the stretch area that must not be stretched.
 6. (option) Choose *Add* in the *Stretch* prompt menu to add previously remove entities to the selection of entities that must be stretched.
 7. Right click to conclude the selection of entities.
The command bar reads: Base point of displacement.
 8. Specify the base point.
The selection stretches dynamically.
The command bar reads: Second point of displacement.
 9. Do one of the following to specify the second displacement point.
 - Click the displacement point.
 - Use Direct Distance Entry: type the displacement distance, then press Enter.
The distance is measured in the cursor direction.
Use ORTHO or POLAR TRACKING to constrain the movement of the cross hairs.
- The selection is stretched.

NOTES To add or remove entities from the selection set in steps 5 and 6 you can use any selection method: picking, window inside or crossing window.

Stretching entities using grips

1. Click the entity you want to stretch.
The entity grips display.
2. Do one of the following:
 - Click a grip to activate it.
The grip is attached to the drawing cursor.
 - Press and hold the Shift key to select multiple grips, then release the Shift key and move one of the selected grips.
3. Click to relocate the grip.
The grip is released from the drawing cursor.

NOTES Use Ortho Mode, Polar Tracking or Snap Tracking to constrain the movement of the drawing cursor.

If you select two (or more) entities with coinciding grips, the shared grips move simultaneously.


Scaling Entities

Command: SCALE

The *Scale* command resizes a selection set in relation to a base point. You can specify the scale factor by selecting a base point and a length or by typing an explicit scale factor in the command bar. The scale factor can also be defined with respect to a base scale factor. E.g. when the base scale factor is 2 and the new scale is 3, the new size is $3/2$ of the original. The base scale and the new scale can also be defined graphically in the drawing.

The Copy option of the Grips Editing commands allows to create multiple copies of the entities being stretched, moved, rotated, mirrored or scaled.


To scale a selection set

1. Do one of the following:
 - Click the *Scale* tool button () on the *Modify* toolbar
 - Choose *Scale* in the *Modify* menu.
 - Type *scale* or *SC* in the command bar.

The command bar reads: Select entities to scale:
2. Select the entities, then right click or press Enter.
The command bar reads: Base point:
3. Specify the base point for the scaling.
The selection set scales dynamically.
The command bar reads: Base scale/<Scale Factor>:
4. Type the scale factor in the command bar and press Enter.
The selection set is scaled.

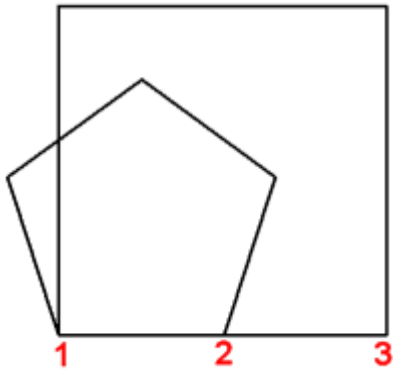
NOTE The length of the vector between the base point and the current cursor position is used as the dynamic scale factor in step 3.

Scaling an entity using a base scale

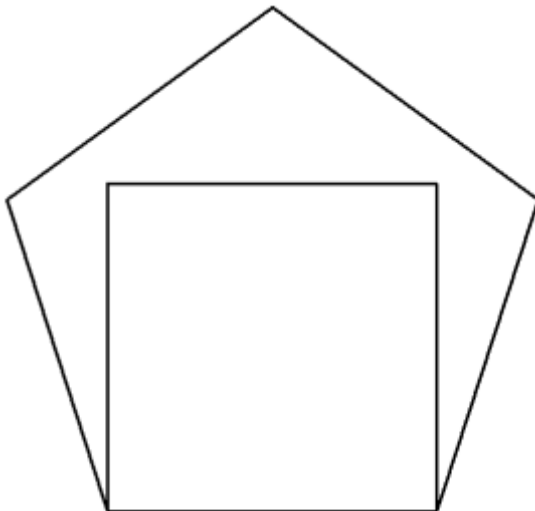
1. Select the entity.
2. Do one of the following:
 - Click the *Scale* tool button () on the *Modify* toolbar
 - Choose *Scale* in the *Modify* menu.
 - Type *scale* or *SC* in the command bar.

The command bar reads: Base point:

3. Specify the base point (1).
The command bar reads: Base Scale/<Scale Factor>:



4. Choose *Base* in the prompt menu of type *B* and press Enter.
The command bar reads: Base scale <1>:
5. Click point 1, then point 2 to define the base scale.
The pentagon scales dynamically.
6. Click point 3.
The edge of the pentagon now equals the side of the square.



Breaking Entities

Command: BREAK


The *Break* command remove a portion of an entity, thus breaking it into two parts.

You can break arcs, circles, ellipses, lines, polylines, rays and infinite lines. Breaking a circle converts it to an arc. A ray is broken into a ray and a line, an infinite line is broken into two rays.

When breaking entities, you must specify two points for the break. By default, the point you use to select the entity becomes the first break point; however, you can use the First option to select a break point different from the one that selects the entity.

To break an entity

1. Do one of the following:

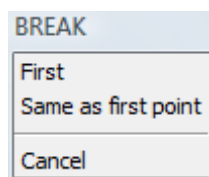
- Click the *Break* tool button () on the *Modify* toolbar.
- Choose *Break* in the *Modify* menu.
- Type *break* or *BR* in the command bar, then press Enter.

The command bar reads: Select entity to break:

2. Click the entity you want to break.

The command bar reads: First break point/<Second break point>:

A prompt menu displays:



3. Click a second point on the entity.

The entity is broken. The portion between the two break points is removed.

Joining Entities

Command: JOIN


The *Join* command joins lines, LW polylines, 2D polylines, 3D polylines, circular arcs, elliptical arcs, splines and helixes at their common endpoints.

The type of the resulting entity depends on the types of the input entities and on their coplanarity.

To join colinear lines

When you join colinear lines, the farthest endpoints remain at their existing locations; BricsCAD draws a new line between these points.

1. Do one of the following:


- Click the *Join* tool button () on the *Modify* toolbar.
- Choose *Join* in the *Modify* menu.
- Type *join* in the command bar, then press Enter.

The command bar reads: Select source entity or multiple entities to join at once:

2. Select the lines.
The command bar reads: Select entities to join:
3. Right click or press Enter.
The lines are joined.

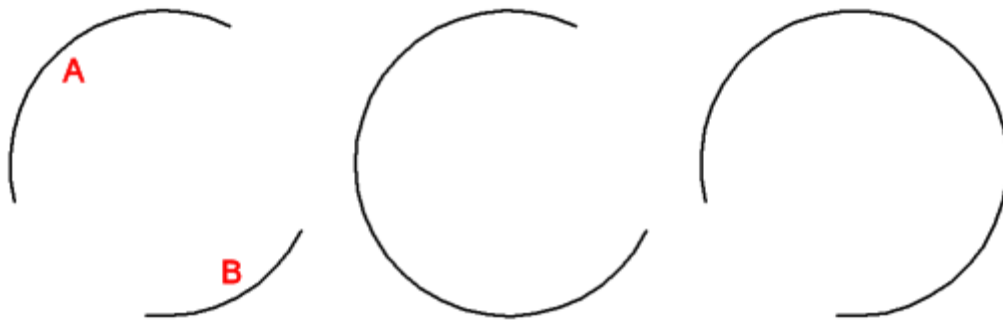
To join two arcs

Coplanar arcs, sharing midpoint and radius, are joined counterclockwise, therefore the result depends on the selecting order.

4. Do one of the following:
 - Click the *Join* tool button () on the *Modify* toolbar.
 - Choose *Join* in the *Modify* menu.
 - Type *join* in the command bar, then press Enter.

The command bar reads: Select source entity or multiple entities to join at once:

5. Select the first arc.
The command bar reads: Select entities to join:
6. Select the second arc and press Enter.
The command bar reads: 2 entities joined into 1 arc



Joining two arcs: start situation (left) and result when selecting A first (middle) or B first (right).

NOTE If more arcs share the same center point and radius, you can select them in step 3. Press Enter after selecting the last arc.

Chamfering and Filletting

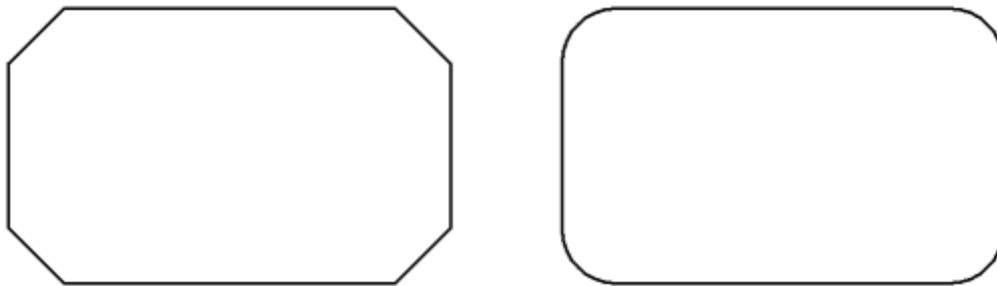
Commands: CHAMFER and FILLET

The *Chamfer* command connects two nonparallel entities with a line to create a beveled edge.

The *Fillet* command connects two entities with an arc of a specified radius to create a rounded edge.

If both entities you are working with are on the same layer, the chamfer or fillet is drawn on that layer. If they are on different layers, the chamfer or fillet is drawn on the current layer.

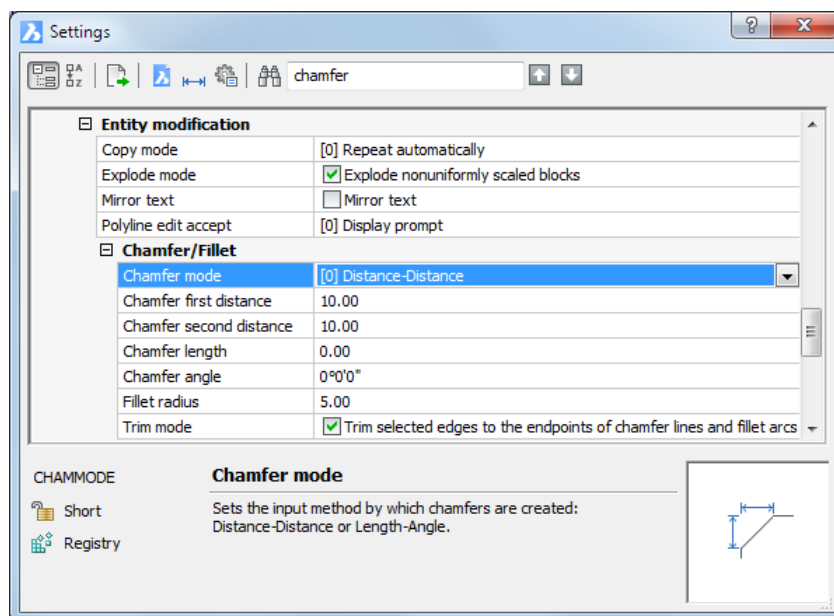
You can choose whether to trim the selected edges to the endpoints of the chamfer lines and fillet arcs or not.



Chamfered rectangle (left) and filleted rectangle (right) with trimmed edges.

Chamfer and fillet settings

1. Open the Settings dialog.
2. Do one of the following:
 - In the *Drawings* settings category, choose *Chamfer/Fillet* in the *Entity Modification* settings sub-category.
 - Type *chamfer* or *fillet* in the search field on the *Settings* dialog window.



Chamfering Entities

Command: CHAMFER

The *Chamfer* command connects two non-parallel entities by extending or trimming them and then joining them with a line to create a beveled edge.

In BricsCAD you can choose between two chamfer methods:


distance-distance: specify how far to trim the entities back from their intersection

distance-angle: specify the length of the chamfer and the angle it forms along the first entity.

The following entities can be chamfered: lines, polylines, rays and infinite lines. When chamfering a polyline, you can create a chamfer between two polyline segments or you can chamfer the entire polyline.

Chamfering using the distance-distance method

1. Do one of the following:

- Click the *Chamfer* tool button () on the *Chamfer/Fillet* flyout of the *Modify* toolbar.
- Choose *Chamfer* in the *Modify* menu.
- Type *chamfer* or *CHA* in the command bar, then press Enter.

The command bar reads: Chamfer (<current chamfer settings>):

Settings/Polyline/Angle/Distance/mEthod/Trim/Undo/Multiple/<Select first entity>:

A prompt menu displays:

CHAMFER
Chamfer Settings...
Polyline
Angle
Distance
mEthod
Trim
Undo
Multiple
Select all entities
Last entity in drawing
Inside window
Crossing window
Outside window
Window polygon
Crossing polygon
Outside polygon
Window circle
Crossing circle
Outside circle
Point
Fence
Cancel

- (option) Choose *Chamfer Settings ...* in the prompt menu or type *S* and press Enter.

In the *Settings* dialog window:

- Specify the *Chamfer first distance*.
- Set the *Chamfer second distance*.
- Set the *Chamfer mode* to *Distance-Distance*.

☐ Chamfer/Fillet	
Chamfer first distance	2.0000
Chamfer second distance	3.0000
Chamfer length	2.5000
Chamfer angle	30
Chamfer mode	Distance-Distance
Fillet radius	0.5000
Trim mode	<input checked="" type="checkbox"/> Trim selected edges to the endpoints of chamfer lines and fillet arcs


Close the *Settings* dialog window.

The command bar reads: Chamfer (<current chamfer settings>):

Settings/Polyline/Angle/Distance/mEthod/Trim/Undo/Multiple/<Select first entity>:

- Select the first entity or polyline segment.
The command window reads: Select second entity.
- Select the second entity or polyline segment.
The chamfer is created.

Chamfering using the length-angle method

- Do one of the following:
 - Click the *Chamfer* tool button () on the *Chamfer/Fillet* flyout of the *Modify* toolbar.
 - Choose *Chamfer* in the *Modify* menu.
 - Type *chamfer* or *CHA* in the command bar, then press Enter.

The command bar reads: Chamfer (<current chamfer settings>):

Settings/Polyline/<Select first entity>:

A prompt menu displays.

- Choose *Chamfer Settings ...* in the prompt menu or type *S* and press Enter.
- In the *Settings* dialog window:
 - Specify the *Chamfer length*.
 - Set the *Chamfer angle*.
 - Set the *Chamfer mode* to *Length-Angle*.


☐ Chamfer/Fillet	
Chamfer first distance	0.5000
Chamfer second distance	0.5000
Chamfer length	2.5000
Chamfer angle	30
Chamfer mode	Length-Angle
Fillet radius	0.5000
Trim mode	<input checked="" type="checkbox"/> Trim selected edges to the endpoints of chamfer lines and fillet arcs

- Close the *Settings* dialog window.
The command bar reads: Chamfer (<current chamfer settings>):
Settings/Polyline/<Select first entity>:
- Select the first entity or polyline segment.
The command window reads: Select second entity.

6. Select the second entity or polyline segment.
The chamfer is created.

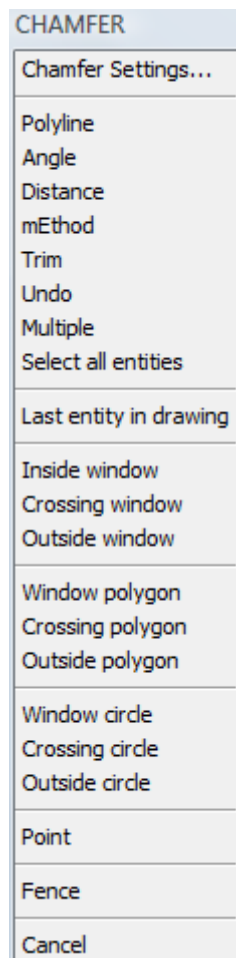
Chamfering all vertices of a polyline

1. Do one of the following:

- Click the *Chamfer* tool button () on the *Chamfer/Fillet* flyout of the *Modify* toolbar.
- Choose *Chamfer* in the *Modify* menu.
- Type *chamfer* or *CHA* in the command bar, then press Enter.

The command bar reads: Chamfer (<current chamfer settings>):
Settings/Polyline/<Select first entity>:

A prompt menu displays:



2. (option) Adjust the *Chamfer Settings*.
3. Choose *Polyline* in the prompt menu or type *P* and press Enter.
The command bar reads: Select 2D polyline to chamfer:
4. Select a polyline.
All vertices of the selected polyline are chamfered.

NOTE When the chamfer method is *distance-angle*, the direction of the polyline defines which is the first entity of a vertex. See drawing rectangles for more information about the direction of closed polylines.

Filleting Entities

Command: FILLET


The *Fillet* command connects two entities with an arc of a specified radius to create a rounded edge.

You can fillet pairs of line segments, straight polyline segments, arcs, circles, rays, and infinite lines. You can also fillet parallel lines, rays, and infinite lines.

When filleting a polyline, you can fillet multiple segments between two selected segments or you can fillet the entire polyline.

Filleting two entities or polyline segments

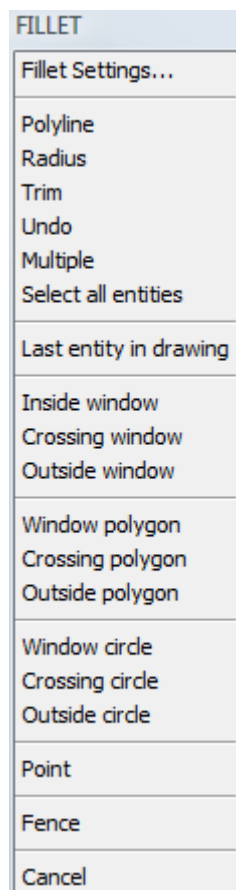
1. Do one of the following:

- Click the *Fillet* tool button () on the *Chamfer/Fillet* flyout of the *Modify* toolbar.
- Choose *Fillet* in the *Modify* menu.
- Type *fillet* or *F* in the command bar, then press Enter.

The command bar reads: Fillet (<current fillet settings>):

Settings/Polyline/Radius/Trim/Undo/Multiple/<Select first entity>:

A prompt menu displays:



2. (option) Choose *Fillet Settings ...* in the prompt menu or type *S* and press Enter. Specify the *Fillet radius* in the *Settings* dialog window:

☐ Chamfer/Fillet	
Chamfer first distance	2.0000
Chamfer second distance	3.0000
Chamfer length	2.5000
Chamfer angle	30
Chamfer mode	Distance-Distance
Fillet radius	2.0000
Trim mode	<input checked="" type="checkbox"/> Trim selected edges to the endpoints of chamfer lines and fillet arcs

Close the Settings dialog window.


The command bar reads: Fillet (<current fillet settings>):

Settings/Polyline/Radius/Trim/Undo/Multiple/<Select first entity>:

3. Select the first entity or polyline segment.
The command window reads: Select second entity.
4. Select the second entity or polyline segment.
The fillet is created.

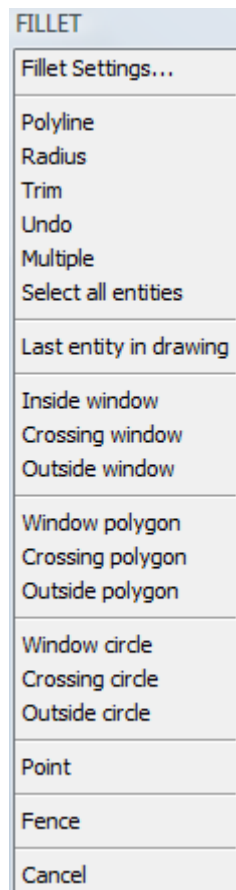
Filleting all vertices of a polyline

1. Do one of the following:

- Click the *Fillet* tool button () on the *Chamfer/Fillet* flyout of the *Modify* toolbar.
- Choose *Fillet* in the *Modify* menu.
- Type *fillet* or *F* in the command bar, then press Enter.

The command bar reads: Fillet (<current fillet settings>): Settings/Polyline/<Select first entity>:


A prompt menu displays:



2. (option) Adjust the *Fillet Settings*.
3. Choose *Polyline* in the prompt menu or type *P* and press Enter.
The command bar reads: Select 2D polyline to chamfer:
4. Select a polyline.

All vertices of the selected polyline are filleted.

Filleting two parallel lines

5. Do one of the following:
 - Click the *Fillet* tool button () on the *Chamfer/Fillet* flyout of the *Modify* toolbar.
 - Choose *Fillet* in the *Modify* menu.
 - Type *fillet* or *F* in the command bar, then press Enter.

The command bar reads: Fillet (<current fillet settings>):
Settings/Polyline/Radius/Trim/Undo/Multiple/<Select first entity>:
6. Select the first entity (line or ray).
The command window reads: Select second entity:
7. Select the second entity (line, ray or infinite line).
The fillet is executed at the endpoint closest to the point where the first entity is selected.
The length of the second entity is adjusted.

Editing Polylines

The PEDIT command modifies any type of two-dimensional or three-dimensional polyline: such as rectangles, polygons and donuts, as well as three-dimensional entities such as pyramids, cylinders and spheres.

Editing a polyline can be:

- opening or closing a polyline;
- changing its overall width or the widths of individual segments;
- converting a polyline with straight line segments into a flowing curve or an approximation of a spline;
- editing individual vertices, adding, removing or moving vertices;
- adding new segments to an existing polyline;
- reversing the direction or order of the vertices;
- merging a series of polylines, lines or arcs, which are connected endpoint to endpoint, into a single polyline;
- turning a line or an arc into a polyline;

To modify a polyline, you first select the polyline, and then select a polyline editing option. The available options vary depending on whether the selected polyline is a two-dimensional or three-dimensional entity.


Converting an entity into a polyline

Command: PEDIT

Lines and arcs can be converted into single-segment polyline.

Converting an entity into a polyline

1. Do one of the following:

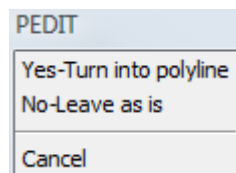
- Click the *Edit Polyline* tool button () on the *Modify* toolbar.
- Choose *Edit Polyline* in the *Modify* menu.
- Type *editpline* or *pedit* in the command window, then press Enter.

The command bar reads: Select polyline to edit.

Select a line or arc.

The command bar reads: The entity selected is not a polyline. Turn it into one? <Y>

A prompt menu displays:



2. Do one of the following:
 - Press Enter to accept the default option.
 - Choose *Yes-Turn into polyline* in the prompt menu.
3. Press Enter to conclude the *Edit Polyline* command.

NOTE The *Explode* tool turns a single segment polyline back into a line or arc.

Opening and closing polylines

Command: PEDIT

When you close a polyline, a straight polyline segment is drawn from the last vertex of the polyline to the first vertex. Opening a polyline removes the closing segment.


When you select a polyline for editing, the prompt menu displays either the *Open* or *Close* option, depending on whether the polyline you select is closed or open.

The following entities are created as closed polylines:

- rectangles
- polygons
- donuts
- revision clouds

You can open or close a polyline using the *Edit Polyline* tool or in the BricsCAD Properties bar.

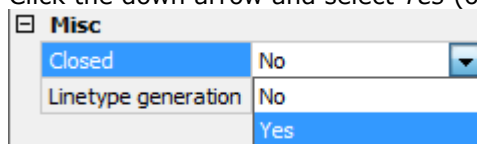
To close or open a polyline

1. Do one of the following:
 - Click the *Edit Polyline* tool button () on the *Modify* toolbar.
 - Choose *Edit Polyline* in the *Modify* menu.
 - Type *editpline* or *pedit* in the command window, then press Enter.

The command bar reads: Select polyline to edit.
2. Click the polyline you want to close or open.
The command bar reads: Edit polyline: Edit vertices/Close (or Open)/Decurve/Fit/Join/Linetype-Mode/Reverse/Spline/Taper/Width/Undo/<eXit> :
A prompt menu displays.
3. To close (or open) a polyline do one of the following:
 - Type *C* (or *O*) in the command bar, then press Enter.
 - Choose *Close* (or *Open*) in the prompt menu.

Opening and closing a polyline using the BricsCAD Properties bar

1. Select the polyline.
The current properties of the selected polyline display in the BricsCAD Properties bar.
2. Select *Closed* in the *Misc* settings category in the *BricsCAD Properties* bar.
3. Click the down arrow and select *Yes* (or *No*).



Joining Polylines

Commands: PEDIT and JOIN

Using the *Join* option of the *Edit Polyline* tool you can add an arc, a line or a polyline entity to an existing open polyline, forming one continuous polyline entity.

To join an entity to a polyline, that entity must already share an endpoint with an end vertex of the selected polyline.


When you join an entity to a polyline, the *width* of the new polyline segment depends on the width of the original polyline and the type of entity you are joining to it:

- A line or an arc inherits the width from the polyline segment to which it is joined.
- A polyline joined to a tapered polyline retains its own width values.
- A polyline joined to a uniform-width polyline inherits the width from the polyline to which it is joined.

The *Join* command joins lines, LW polylines, 2D polylines, 3D polylines, circular arcs, elliptical arcs, splines and helixes at their common endpoints.

The type of the resulting entity depends on the types of the input entities and on their coplanarity.

To join an arc, line, or polyline to an existing polyline

1. Do one of the following:
 - Click the *Edit Polyline* tool button () on the *Modify* toolbar.
 - Choose *Edit Polyline* in the *Modify* menu.
 - Type *editpline* or *pedit* in the command window, then press Enter.

The command bar reads: Select polyline to edit.
2. Select the parent polyline.
The command bar reads: Edit polyline: Edit vertices/Close (or Open)/Decurve/Fit/Join/Linetype-Mode/Reverse/Spline/Taper/Width/Undo/<eXit>:
A prompt menu displays.
3. Choose *Join* in the prompt menu or type *J* and press Enter.
The command bar reads: Select entities to join:
4. Select the entities to join.
The number of already selected entities displays in the command bar.
5. Right click or press Enter to stop selecting entities and add the selection to the polyline.
6. Do one of the following:
 - Continue editing the selected polyline.
 - Choose *Exit* in the prompt menu or press Enter to conclude the *Edit Polyline* tool.


NOTE You can select multiple entities in step 3 on condition they form one chain with the parent polyline.

Changing the polyline width

Command: PEDIT


The *Width* option of the *Edit Polyline* tool applies a uniform width to the entire entity, while the *Taper* option tapers the polyline uniformly along its entire length.

To apply a uniform width to a polyline

1. Do one of the following:
 - Click the *Edit Polyline* tool button () on the *Modify* toolbar.
 - Choose *Edit Polyline* in the *Modify* menu.
 - Type *editpline* or *pedit* in the command window, then press Enter.

The command bar reads: Select polyline to edit.
2. Select the polyline.
The command bar reads: Edit polyline: Edit vertices/Close (or Open)/Decurve/Fit/Join/Linetype-Mode/Reverse/Spline/Taper/Width/Undo/<eXit>:
A prompt menu displays.
3. Choose *Width* in the prompt menu or type *W* and press Enter.
The command bar reads: Enter new width for all segments:
4. Type a new width in the command bar and press Enter.
The new width is applied to the entire polyline.
5. Do one of the following:
 - Continue editing the selected polyline.
 - Choose *Cancel* in the prompt menu or press Enter or right click to conclude the *Edit Polyline* tool.

To taper a polyline uniformly along its length

1. Do one of the following:
 - Click the *Edit Polyline* tool button () on the *Modify* toolbar.
 - Choose *Edit Polyline* in the *Modify* menu.
 - Type *editpline* or *pedit* in the command window, then press Enter.

The command bar reads: Select polyline to edit.
2. Select the polyline.
The command bar reads: Edit polyline: Edit vertices/Close (or Open)/Decurve/Fit/Join/Linetype-Mode/Reverse/Spline/Taper/Width/Undo/<eXit>:
A prompt menu displays.
3. Choose *Taper* in the prompt menu or type *T* and press Enter.
The command bar reads: Enter new starting polyline width <current starting width>:
4. Type a new starting width in the command bar and press Enter.
The command bar reads: Enter new ending polyline width <current ending width>:
5. Type a new ending width in the command bar and press Enter.
The polyline is tapered uniformly along its length.
6. Do one of the following:
 - Continue editing the selected polyline.
 - Choose *Cancel* in the prompt menu or press Enter or right click to conclude the *Edit Polyline* tool.

Editing polyline vertices

Commands: PEDIT and PEDITTEXT

The *Edit vertices* option of the *Pedit* command modifies individual polyline vertices.

When you select this option, the program switches into a special vertex editing mode and places an X on the first vertex. The *Next* and *Previous* options move the X to the next or previous vertex. You can edit only one vertex at a time.

When editing vertices, you can modify the polyline in the following ways:

Convert a straight polyline segment into a curve.

Break a polyline into two separate polylines.

Insert a new vertex in a polyline.

Move a vertex in a polyline.

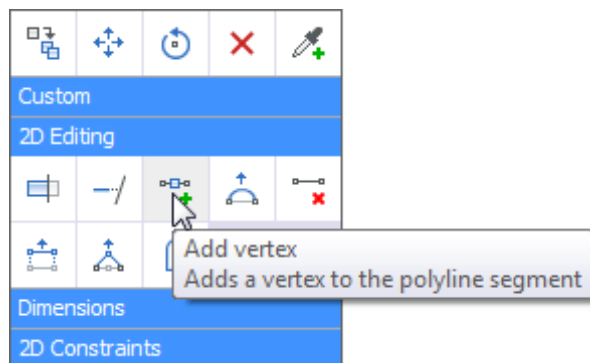
Delete vertices in a polyline.







Change the width of a polyline segment.

The *Peditext* command is used in *2D Editing* command group of the Quad cursor menu

Editing a polyline using the Quad


1. Move the cursor over the polyline.
2. When the Quad cursor menu displays, do one of the following:
 - Right click to repeat the previous *Quad* command
 - Move the cursor to the *Quad* and move to the *2D Editing* command group.



3. Select a command:
 -  Add vertex: adds a vertex to the segment under the cursor.
 -  Add vertex at end: adds as a new vertex at the end of the polyline.
This option is only available if the cursor is on one of the endpoints of a polyline.
 -  Adjust Bulge: Modifies the bulge factor of the segment under the cursor.
 -  Delete segment: deletes the segment under the cursor.
 -  Stretch segment: moves the segment under the cursor.
 -  Stretch vertex: moves the vertex closest to the cursor.

Starting the polyline vertex editing mode

1. Do one of the following:

- Click the *Edit Polyline* tool button () on the *Modify* toolbar.
- Choose *Edit Polyline* in the *Modify* menu.
- Type *editpline* or *pedit* in the command window, then press Enter.

The command bar reads: Select polyline to edit.

2. Select the polyline.

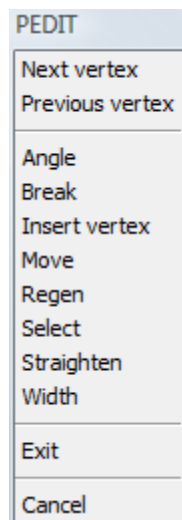
The command bar reads: Edit polyline: Edit vertices/Close (or Open)/Decurve/Fit/Join/Linetype-Mode/Reverse/Spline/Taper/Width/Undo/<eXit>:
A prompt menu displays.

3. Choose *Edit vertices* in the prompt menu or type *E* and press Enter.

The command bar reads:

Next/Previous/Angle/Break/Insert/Move/Regen/SElect/Straighten/Width/eXit/<Next>:

The *Editpline* prompt menu changes:



A triangle indicates the start point of the polyline. The first vertex of the polyline is selected.

Convert a straight polyline segment into an arc

1. Start the polyline vertex editing mode.
2. Select the start vertex of the segment you want to convert.
Choose *Next vertex* / *Previous vertex* in the prompt menu to select a vertex.
The *X* indicates the currently selected vertex.
3. Choose *Angle* in the prompt menu or type *A* and press Enter.
The prompt menu closes.
The command bar reads: Included angle for segment (>0 is ccw, 0 is straight, <0 is cw) <current angle>:
4. Type a new included angle for the segment and press Enter.
The direction of an arc segment is as follows:
 - positive angles: counter clockwise
 - negative angle: clockwise direction
 - 0° for straight segments
5. Choose *Exit* in the prompt menu or type *X* and press Enter to leave the *Polyline vertex editing mode*.

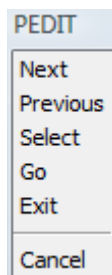
6. Do one of the following:
 - Continue editing the selected polyline.
 - Choose *Exit* in the prompt menu or press Enter to conclude the *Edit Polyline* tool.

Break a polyline into two separate polylines

1. Start the polyline vertex editing mode.
2. Select the start vertex where you want to break the polyline.
Choose *Next vertex / Previous vertex* in the prompt menu to select a vertex.
The X indicates the currently selected vertex.
3. Choose *Break* in the prompt menu or type *B* and press Enter.

The command bar reads: Next/Previous/Select/Go/eXit/<Next>:

The prompt menu changes:



4. (option) Do one of the following to select a second vertex:
 - Choose *Next vertex / Previous vertex* in the prompt menu.
 - Choose *Select* in the prompt menu or type *S* and press Enter.
You are prompted to select a vertex.
5. Choose *Go* in the prompt menu or type *G* and press Enter.
The polyline s broken into two polylines.
If a second point is selected in step 4 the segment(s) between the selected vertices is (are) deleted.
6. Choose *Exit* in the prompt menu or type *X* and press Enter to leave the *Polyline vertex editing mode*.
7. Do one of the following:
 - Continue editing the selected polyline.
 - Choose *Exit* in the prompt menu or press Enter to conclude the *Edit Polyline* tool.

Insert a new vertex in a polyline

1. Start the polyline vertex editing mode.
2. Select the start vertex of the segment where you want to insert a vertex.
Choose *Next vertex / Previous vertex* in the prompt menu to select a vertex.
The X indicates the currently selected vertex.
3. Choose *Insert vertex* in the prompt menu or type *I* and press Enter.
The command bar reads: Location for new vertex:
4. Specify the location for the new vertex.
The new vertex is inserted.
5. Choose *Exit* in the prompt menu or type *X* and press Enter to leave the *Polyline vertex editing mode*.

6. Do one of the following:
 - Continue editing the selected polyline.
 - Choose *Exit* in the prompt menu or press Enter to conclude the *Edit Polyline* tool.

Move a vertex in a polyline

1. Start the polyline vertex editing mode.
2. Select the vertex you want to move.
Choose *Next vertex / Previous vertex* in the prompt menu to select a vertex.
The *X* indicates the currently selected vertex.
3. Choose *Move* in the prompt menu or type *M* and press Enter.
The command bar reads: New location for vertex.
4. Specify the new location for the selected vertex.
The vertex is moved.
5. Choose *Exit* in the prompt menu or type *X* and press Enter to leave the *Polyline vertex editing mode*.
6. Do one of the following:
 - Continue editing the selected polyline.
 - Choose *Exit* in the prompt menu or press Enter to conclude the *Edit Polyline* tool.

Delete vertices in a polyline

1. Start the polyline vertex editing mode.
2. Select the vertex before the first vertex you want to delete.
Choose *Next vertex / Previous vertex* in the prompt menu to select a vertex.
The *X* indicates the currently selected vertex.
3. Choose *Straighten* in the prompt menu or type *S* and press Enter.
The command bar reads: Straighten: Next/Previous/Select/Go/eXit/<Next>:
4. Do one of the following to select the vertex after the last vertex you want to delete:
 - Choose *Next vertex / Previous vertex* in the prompt menu.
 - Choose *Select* in the prompt menu or type *S* and press Enter.
You are prompted to select a vertex.
5. Choose *Go* in the prompt menu or type *G* and press Enter.
The vertices between the selected vertices are removed.
A straight segment is drawn between the selected vertices.
6. Choose *Exit* in the prompt menu or type *X* and press Enter to leave the *Polyline vertex editing mode*.
7. Do one of the following:
 - Continue editing the selected polyline.
 - Choose *Exit* in the prompt menu or press Enter to conclude the *Edit Polyline* tool.

Change the width of a polyline segment

1. Start the polyline vertex editing mode.
2. Select the start vertex of the segment you want to change the width of.
Choose *Next vertex / Previous vertex* in the prompt menu to select a vertex.
The *X* indicates the currently selected vertex.
3. Enter starting width <current width>:
4. Do one of the following:

- Type the new width in the command bar and press Enter.
- Click to define the width graphically.

The command bar reads: Enter ending width <current width>:

- Do one of the following:
 - Type the new width in the command bar and press Enter.
 - Click to define the width graphically.
- Choose *Exit* in the prompt menu or type *X* and press Enter to leave the *Polyline vertex editing mode*.
- Do one of the following:
 - Continue editing the selected polyline.
 - Choose *Exit* in the prompt menu or press Enter to conclude the *Edit Polyline* tool.

NOTE The new width is applied when you conclude the *Edit Polyline* tool in step 6.


Curving and decurving polylines

Command: PEDIT

The *Fit* or *Spline* options of the *Edit Polyline* tool convert a multi-segment polyline into a smooth curve. The *Fit* option creates a smooth curve connecting all the vertices. The *Spline* option computes a smooth curve that is pulled toward the vertices but passes through only the first and last vertices.

The *Decurve* option removes *Fit* or *Spline* curves and arcs, leaving straight segments between the vertices.

To fit a curve to a polyline

- Do one of the following:
 - Click the *Edit Polyline* tool button () on the *Modify* toolbar.
 - Choose *Edit Polyline* in the *Modify* menu.
 - Type *editpline* or *pedit* in the command window, then press Enter.

The command bar reads: Select polyline to edit.

- Select the polyline.
The command bar reads: Edit polyline: Edit vertices/Close (or Open)/Decurve/Fit/Join/Linetype-Mode/Reverse/Spline/Taper/Width/Undo/<eXit>:
A prompt menu displays.
- Do one of the following:
 - Choose *Fit* in the prompt menu or type *F* and press Enter.
 - Choose *Spline* in the prompt menu or type *S* and press Enter.



Original polyline

After applying Fit

After applying Spline

4. Do one of the following:
 - Continue editing the selected polyline.
 - Choose *Done* in the prompt menu or press Enter to conclude the *Edit Polyline* tool.


NOTES Use the *Decurve* option in step 3 of the above procedure to restore the original polyline. The *SPLINESEGS* (Spline Segments) settings variable controls the number of segments (lines or arcs) to be generated for spline-fit polylines.

Setting the Linetype generation mode

Command: PEDIT

The *Linetype mode* option of the *Edit Polyline* tool lets you change the way how a dashed linetype is applied to a multi-segment polyline.

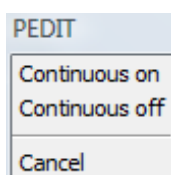
To set the Linetype mode

1. Do one of the following:
 - Click the *Edit Polyline* tool button () on the *Modify* toolbar.
 - Choose *Edit Polyline* in the *Modify* menu.
 - Type *editpline* or *pedit* in the command window, then press Enter.

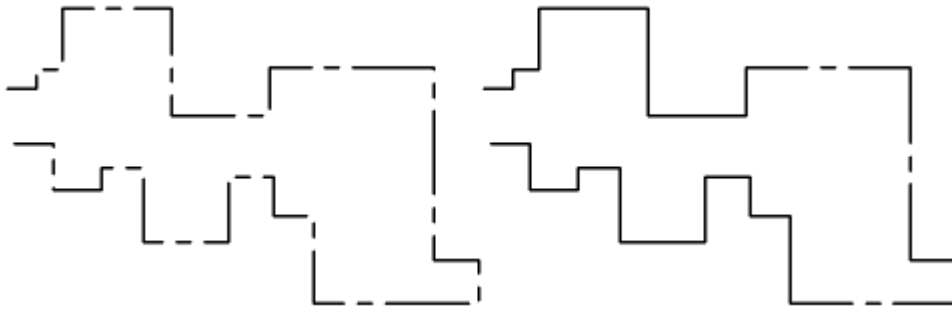
The command bar reads: Select polyline to edit.
2. Select the polyline.

The command bar reads: Edit polyline: Edit vertices/Close (or Open)/Decurve/Fit/Join/Linetype-Mode/Reverse/Spline/Taper/Width/Undo/<eXit>:
A prompt menu displays.
3. Choose *Linetype* mode in the prompt menu or type *L* and press Enter.

The command bar reads: Linetype continuous along polyline: ON/OFF <current setting>:
The *Editpline* prompt menu changes:



4. Do one of the following:
 - Choose *Continuous on* or *Continuous off* in the prompt box.
 - Type *ON* or *OF* in the command bar, then press Enter.



The same polyline with Continuous On (left) and Continuous Off (right)

5. Do one of the following:
 - Continue editing the selected polyline.
 - Choose *Cancel* in the prompt menu or press Enter or right click to conclude the *Edit Polyline* tool.

Editing polylines in the Properties bar

When you select polyline, its current properties display in the BricsCAD Properties bar. In the *BricsCAD Properties bar* the following polyline properties can be edited:

- Move vertices (1)
- Change the width of polyline segments (2)
- Convert straight segments into a curve (3)
- Change the global width (4)
- Change the Elevation (5)
- Open / close the polyline (6)
- Change the Linetype generation mode (7)

Polyline	
General	
Color	<input type="checkbox"/> ByLayer
Layer	0
Linetype	———— ByLayer
Linetype scale	1
Plot style	ByColor
Lineweight	———— ByLayer
Hyperlink	
Thickness	0
3D Visualization	
Material	ByLayer
Geometry	
Vertices	8
Vertex	1
1 Position	-481, 1536
	X -481
	Y 1536
2 Start width	0
	End width 0
3 Bulge	0
4 Global width	0
5 Elevation	0
	Area 2423602
	Length 8469
Misc	
6 Closed	No
7 Linetype generation	Disabled

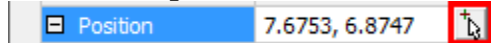
To move polyline vertices

1. Select the polyline.
2. Click the *Vertex* field in the *Properties bar*, then press the arrow buttons to select the vertex you want to move.

A X indicates the currently selected vertex in the drawing.

Vertex	2	◀ ▶
--------	---	-----

3. Do one of the following:
 - Adjust the *X-* and/or *Y-field* and press Enter.
 - Click the *Vertex Position* field and press the *Position* button to move the vertex in the drawing.



4. Continue editing the polyline or press the Esc key to stop.

To change the width of polyline segments

1. Select the polyline.
2. Click the *Vertex* field in the *Properties bar*, then press the arrow buttons to select the start vertex of the segment you want to modify.

A *X* indicates the currently selected vertex in the drawing.

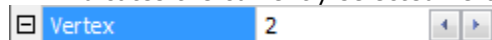


3. Type a new width in the *Start width* field and press Enter.
4. Type a new width in the *End width* field and press Enter.
5. Continue editing the polyline or press the Esc key to stop.

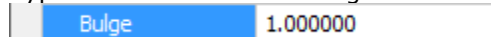
To convert straight segments into a curve

1. Select the polyline.
2. Click the *Vertex* field in the *Properties bar*, then press the arrow buttons to select the start vertex of the segment you want to modify.

A *X* indicates the currently selected vertex in the drawing.

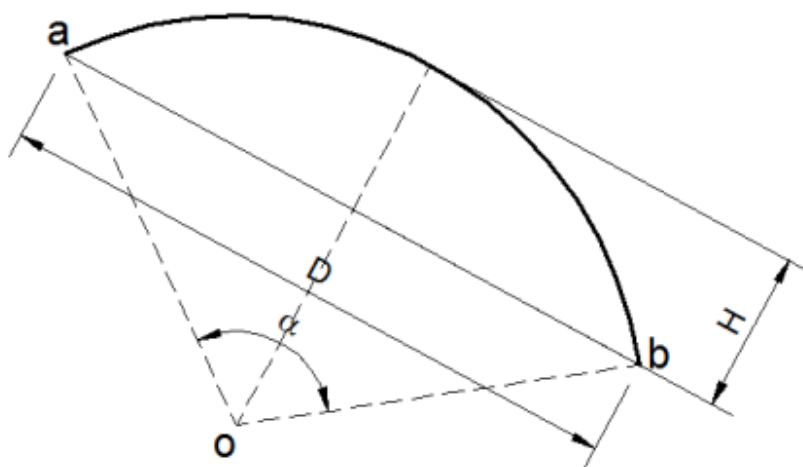


3. Type a new value in the *Bulge* field and press Enter.



4. Continue editing the polyline or press the Esc key to stop.

NOTE The bulge is the tangent of one fourth the included angle for an arc segment. A positive bulge creates a counter clockwise arc, while a negative bulge creates an arc with a clockwise direction. Straight segments have a zero bulge.



$$\text{Bulge} = 2H/D$$

$$\text{Bulge} = \tan \alpha / 4$$

Converting Entities

The Explode tool converts complex entities, such as blocks, polylines, solids or dimensions into their component parts.

The Region tool converts closed entities into a solid entity.

The Flatten command converts splines to polylines.

Exploding entities

Commands: EXPLODE and TXTEXP

The *Explode* command converts complex entities, such as blocks, polylines, solids or dimensions into their component parts.

Exploding a polyline or dimension reduces it to a collection of individual line and arc entities that you can then modify individually.

Blocks are converted to the individual entities, possibly including other, nested blocks that composed the original entity.


Multiline texts explode into single line text entities.

In general exploding entities will have no visible effect in the drawing, except for:

- If the original polyline had a width, the width information is lost when you explode it. The resulting lines and arcs follow the centerline of the original polyline.
- If you explode a block containing attributes, the attributes are lost, but the original attribute definitions remain.
Colors and linetypes assigned *BYBLOCK* may appear different after exploding an entity, because they will adopt the default color and linetype.

The *TxtExp* command explodes text into polylines (short for "text explode").

To explode entities

1. Do one of the following:
 - Click the *Explode* tool button () on the *Modify* toolbar.
 - Choose *Explode* in the *Modify* menu.
 - Type *explode* in the command bar, then press Enter.

The command window reads: Select entities to explode.

2. Select the entities, then right click or press Enter.

NOTE If you select the entities first, then launch the *Explode* tool, the selected entities are exploded immediately.

Creating Regions


Command: REGION

The *Region* command converts closed entities into a solid entity.

You can create regions from closed entities, such as polylines, polygons, circles, ellipses, closed splines and donuts.

Creating regions typically has no visible effect on a drawing. However, if the original entity had a width or lineweight, that information is lost when you create the region.

To create regions

1. Do one of the following:
 - Click the *Region* tool button () in the *Draw* toolbar.
 - Choose *Region* in the *Draw* menu.
 - Type *region* in the command bar, then press Enter.

The command bar reads: Select objects:
2. Select the entities, then right click or press Enter.

The command bar displays how many regions have been created.

Measuring and Dividing Entities

The Measure tool places markers - points or blocks - at a specified interval along the length or circumference of an entity.

The Divide tool places markers - points or blocks - along a selected entity. The markers evenly divide the entity into the specified number of equal parts.

Measuring Entities


Command: MEASURE

The *Measure* command places markers - points or blocks - at a specified interval along the length or circumference of an entity. The Measure tool starts placing markers at the closes endpoint to where you select the entity.

You can measure lines, polylines, arcs, circles, ellipses, elliptical arcs and splines. You cannot measure rays and infinite lines.

You measure a circle along its circumference starting from the *Angle Base* value as defined in the *Settings* dialog. If *Angle Base* is set to zero, a circle is measured starting at 3 o'clock (east).

To measure an entity using points

1. Do one of the following:
 - Click the *Measure* tool button () on the *Modify* toolbar.
 - Choose *Measure* in the *Modify* menu.
 - Type *measure* in the command bar, then press Enter.


The command bar reads: Select entity to measure:

2. Click the entity you want to measure.
The command bar reads: Block/ <Segment length>:
3. To define the measurement interval, do one of the following:
 - Type the measurement interval and press Enter.
 - Specify two points in the drawing.

Points are placed along the selected entity at the specified interval.

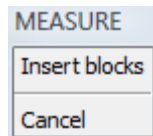
NOTE It might be necessary to adjust the Point display mode settings to display the points correctly.

To place blocks a specified interval along an entity

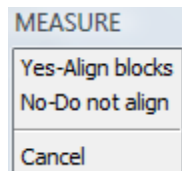
1. Do one of the following:
 - Click the *Measure* tool button () on the *Modify* toolbar.
 - Choose *Measure* in the *Modify* menu.
 - Type *measure* in the command bar, then press Enter.

The command bar reads: Select entity to measure:

2. Click the entity you want to measure.
The command bar reads: Block/ <Segment length>:
A prompt menu displays:



3. Choose *Insert blocks* in the prompt menu or type B and press Enter.
The command bar reads: Name of block to insert:
4. Type the name of the block in the command bar and press Enter.
The command bar reads: Align blocks with entity? <Y>:
A prompt menu displays:

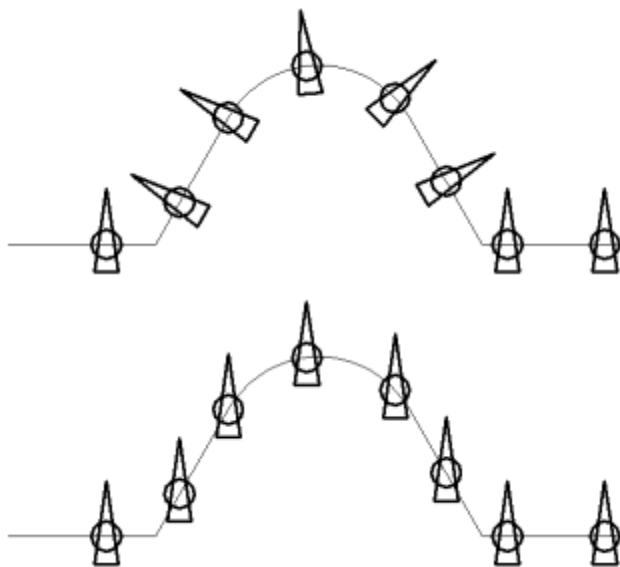


5. Do one of the following:
 - Choose *Yes-Align blocks* in the prompt menu or press Enter to align the blocks with the selected entity.
 - Choose *No-Do not align* in the prompt menu or type N and press Enter to place the blocks not rotated.

The command bar reads: Segment length:

6. To define the measurement interval, do one of the following:
 - Type the measurement interval and press Enter.
 - Specify two points in the drawing.

The blocks are placed along the selected entity at the specified interval.



Blocks aligned (top) or not aligned (bottom)

7.


Dividing Entities

Command: DIVIDE

The *Divide* command places markers - points or blocks - along a selected entity. The markers evenly divide the entity into the specified number of equal parts.

You can divide lines, polylines, arcs, circles, ellipses, elliptical arcs and splines.


To divide an entity using points

1. Do one of the following:
 - Click the *Divide* tool button () on the *Modify* toolbar.
 - Choose *Divide* in the *Modify* menu.
 - Type *divide* in the command bar, then press Enter.

The command bar reads: Select entity to divide:
2. Click the entity you want to divide.
The command bar reads: Blocks/<Number of Segments>:
3. Type the number of segments in the command bar and press Enter.
Points are placed along the selected entity to divide the entity.

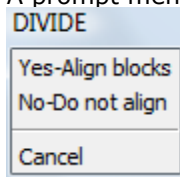
NOTE It might be necessary to adjust the Point display mode settings to display the points correctly.

To divide an entity using blocks

1. Do one of the following:
 - Click the *Divide* tool button () on the *Modify* toolbar.
 - Choose *Divide* in the *Modify* menu.
 - Type *divide* in the command bar, then press Enter.

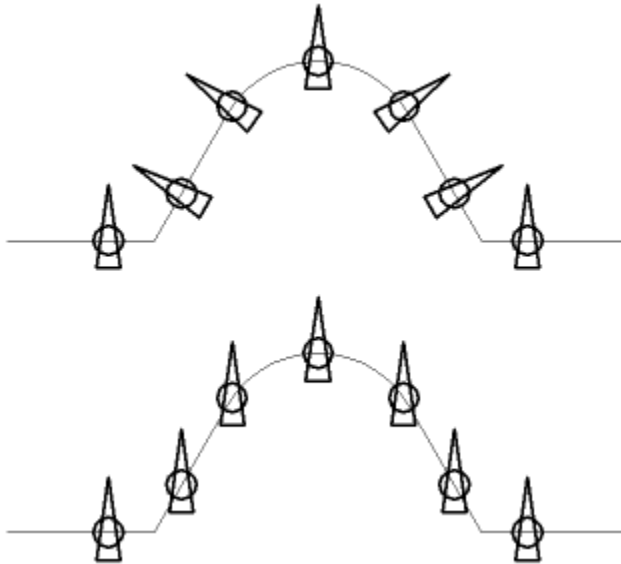
The command bar reads: Select entity to divide:
 2. Click the entity you want to divide.
The command bar reads: Blocks/<Number of Segments>:
A prompt menu displays:
-
3. Choose *Insert blocks* in the prompt menu or type B and press Enter.
The command bar reads: Name of block to insert:

4. Type the name of the block in the command bar and press Enter.
The command bar reads: Align blocks with entity? <Y>:
A prompt menu displays:



5. Do one of the following:
- Choose *Yes-Align blocks* in the prompt menu or press Enter to align the blocks with the selected entity.
 - Choose *No-Do not align* in the prompt menu or type N and press Enter to place the blocks not rotated.

The blocks are placed along the selected entity to divide the entity.



Blocks aligned (top) or not aligned (bottom)

Blocks, attributes and external references

Blocks, attributes, and external references provide mechanisms for managing entities in your drawings and for including additional information with the standard drawing entities.

Blocks

Blocks allow you to combine numerous entities into a single entity and reuse it, inserting multiple copies. A block definition is only saved once in a drawing. If a block definition is redefined, all instances of the block will be updated automatically. Block definitions can be copied between drawings.

Attributes

With attributes, you can associate text, such as part numbers or prices, with blocks and then extract the text-attribute information to a separate file, such as a database, for further analysis.

External references

With external references, you can link separate reference drawing files to the current drawing to combine information without adding the contents of the reference drawings to a drawing. If you make changes to the referenced file, all references are updated automatically.

Blocks



Working with Blocks (Internet connection needed)

A block is a single entity which consists of visible entities such as lines, arcs, circles, text, dimensions, etc.

Apart from visible entities a block can also contain invisible data called *attributes*. Blocks are stored as part of the drawing file.

Blocks can help you to better organize your work, quickly create and revise drawings and reduce the drawing file size.

- Using blocks, you can create a library of frequently used symbols. Then you can insert a symbol as a block rather than redraw the symbol from scratch.
- You can insert multiple instances of a single block definition. You can change the block definition to quickly revise a drawing, and then update all instances of the block.

A procedure called *nesting* is used to combine blocks in a new block. Nesting is useful when you want to combine and include small components, such as nuts and bolts, into a larger assembly and you need to insert multiple instances of that assembly into an even larger drawing.

When you insert a block, the following rules apply to define the color, linetype and lineweight of the entities in the block:

- Entities which are created on layers other than layer 0 or with explicitly specified color, linetype and lineweight, retain their original settings.
- Entities which are created on layer 0 and color, linetype and lineweight *BYLAYER*: are inserted on the current layer and adopt the color, linetype and lineweight of the insertion layer.
- Entities which originally have their color, linetype and lineweight assigned *BYBLOCK*, adopt their properties of the block.

Creating Blocks

Commands: BLOCK, -BLOCK, WBLOCK, PASTEBLOCK and SECTIONPLANETOBLOCK

The *Block* command groups entities into a block using a dialog box. The *-Block* commands prompts you in the command bar.

The *Wblock* command writes blocks and other drawing parts to a separate drawing file (short for "write block").


The *Pasteblock* command pastes entities from the *Clipboard* as a block in the current drawing (see Paste a selection as a block).

The *SectionPlaneToBlock* command saves the selected section plane to a 2D cross section / elevation block or a 3D cutaway section block.

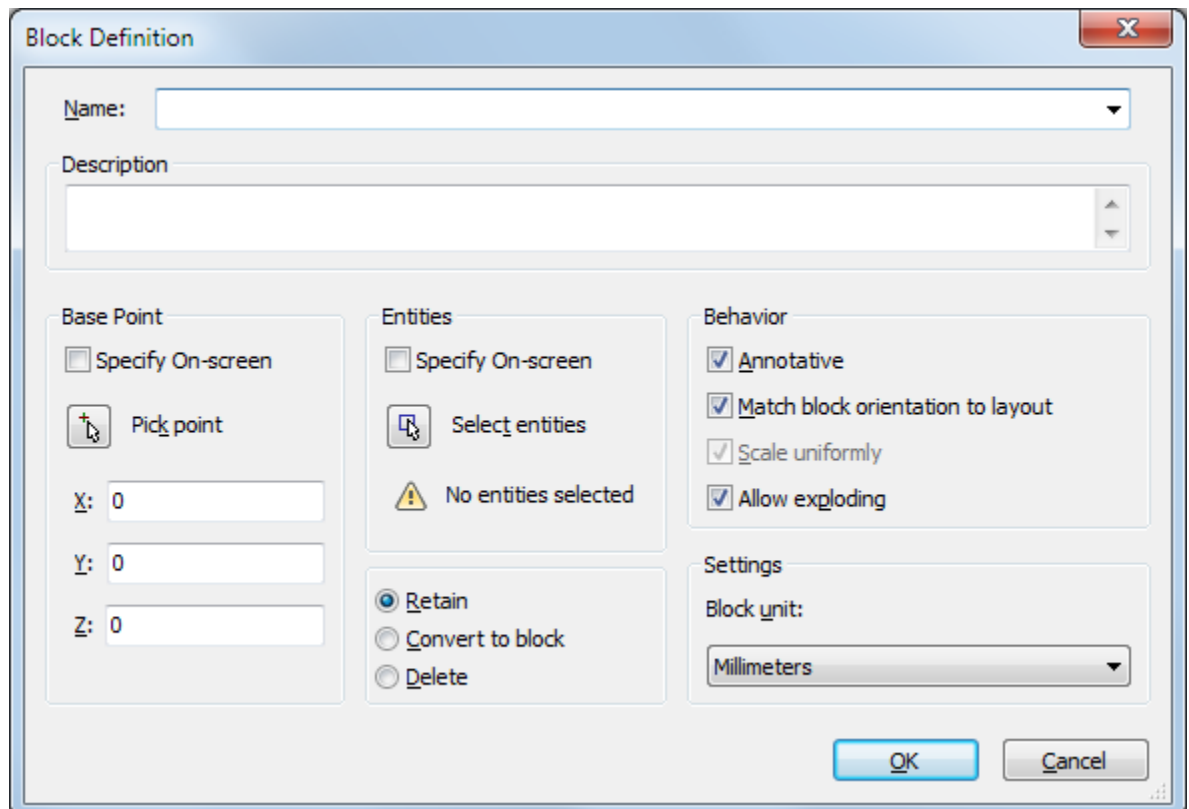
General procedure to create a block



1. Specify the name.
2. Specify the insertion point.
The insertion point is the base point for the block and serves as the reference point when you later insert the block into a drawing.
3. Select entities.

To create a block in a drawing

1. Do one of the following:
 - Click the *Create Block* tool button () on the *Tools* toolbar.
 - Choose *Create Block* in the *Tools* menu.
 - Type *block* in the command bar, then press Enter.

The *Block Definition* dialog displays:




2. Type a name for the new block in the *Name* field.
3. Do one of the following to specify the *Base Point*:
 - Check the *Specify On-screen* option.
You will be prompted to specify the base point after clicking the *OK* button.
 - Click the *Pick point* button ()
The *Block Definition* dialog closes temporarily to let you pick a point in the drawing.
 - Type the coordinates of the base point in the X-, Y- and Z-field.
4. Do one of the following to select the entities to be included in the block definition:
 - Check the *Specify On-screen* option.
You will be prompted to select the entities after clicking the *OK* button.
 - Click the *Select entities* button ()
The *Block Definition* dialog closes temporarily to let select the entities in the drawing.
5. Set the block properties:
 - *Annotative*: creates an annotative the block.
 - *Match block orientation to layout*: if checked, the orientation of an annotative block is preserved in a rotated viewport.
 - *Scale uniformly*: if checked, the X-, Y- and Z-scaling cannot be set differently.
 - *Allow exploding*: if not checked, prevents the block from being exploded.
 - *Block unit*: set the insertion unit for the block (see also: To insert a block).
6. Set the behavior of the selected entities after the block is created:
 - *Retain*: The selected entities remain in the drawing.
 - *Convert to block*: The selected entities are replaced by the newly created block.
 - *Delete*: The selected entities are deleted.
7. Click the *OK* button to create the block.

NOTE | If the *Delete* option in step 6 was set, the OOPS command brings back the erased block entities.

To create a block using the Blocks Explorer

1. Do one of the following:
 - *Choose Blocks...* in the *Settings* menu.
 - *Type explblocks* in the command bar, then press *Enter*.


The Drawing Explorer - Blocks dialog opens.
2. In the *Drawing Explorer* dialog do one of the following:
 - Click the *New* button () on the *Details* toolbar.
 - Choose *New* in the *Edit* menu.

The *Drawing Explorer* dialog closes.

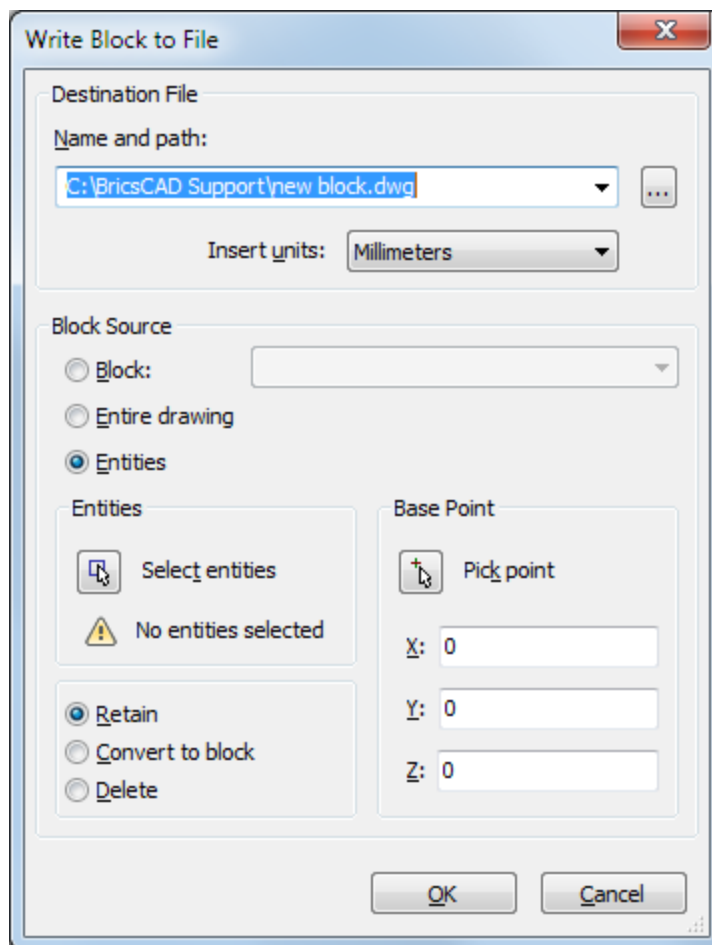
The command bar reads: Insertion point for new block:
3. Specify the insertion point of the new block.
The command bar reads: Select entities for block.
4. Select the entities for the new block.
5. Right click to create the block.
The selected entities are removed from the drawing.
The *Drawing Explorer* dialog reopens.
The *Name* field of the new block is selected.
6. Type a name in the *Name* field, replacing the default *NewBlock1* name.



7. Close the *Drawing Explorer* dialog.
8. (option) Type *oops* in the command bar, then press Enter to bring back the selected block entities.

To save a block to a separate file


1. Do one of the following:
 - Click the *Save Block...* tool button () on the *Tools* toolbar.
 - Choose *Save Block...* in the *Tools* menu.
 - Type *wblock* in the command bar, then press Enter.

The *Write Block to File* dialog opens.



2. Click the browse button () next to the *Name and path* field. The *Save Block* dialog opens.
3. On the *Save Block* dialog, select a folder and type a name in the *File name* field, then click the *Save* button. The *Save Block* dialog closes.
4. *Insert units*: set the insertion unit for the block (see also: To insert a block).
5. To select the source entities for the new block, do one of the following:
 - Click the *Block* radio button, then select a block from the drop-down list.
 - Click the *Model* space radio button, to select all the entities in the model space of the current drawing.
 - Click the *Entities* radio button, then click the *Select Entities* button (). The *Write Block to File* dialog temporarily closes to let you select entities in the drawing.

Right click to stop selecting entities.

Click the *Pick point* button () to define the *Base point* of the new drawing or

Type the coordinates of the base point in the X-, Y- and Z-field.

Set the behavior of the selected entities after the block is created:

- *Retain*: The selected entities remain in the drawing.
- *Convert to block*: The selected entities are replaced by the newly created block.
- *Delete*: The selected entities are deleted.

6. Click the *OK* button to create the drawing file.

NOTE | If the *Delete* option in step 5 was set, the OOPS command brings back the erased block entities.

Inserting Blocks

Commands: INSERT, -INSERT, INSERTALIGNED and MININSERT

The *Insert* command lets you insert blocks through a dialog box.

The *-Insert* command inserts blocks by prompting in the command bar. BricsCAD will lookup the block name in the block definitions in the current drawing. If the block is not found in the current drawing, the paths defined by the *SRCHPATH* system variable are searched. If the block is not found there either, BricsCAD responds 'Could not find file <blockname>'

The *Insertaligned* command inserts a block entity, with easy alignment on existing entities. The *Insertaligned* command is similar to the *Insert* command, but during the placement the block will be dynamically aligned with existing lines, polylines, arcs or circles. The block is aligned with the entity to which the cursor snaps. If no entity is snapped, the block is aligned with the X-axis of the WCS.

The *Mininsert* command Inserts a block as a rectangular array; combines the *-Insert* and *Array* commands (short for "multiple insertion").

You can choose to select an existing block definition or to insert an entire drawing as a block. When you insert a drawing, a new block definition is created in the current drawing. If you change the original drawing file, those changes have no effect on the current drawing unless you redefine the block by reinserting the changed drawing. When a block is inserted in a drawing, it is treated as a single entity.

General procedure to insert a block:

1. Specify a block definition or drawing file.
2. Specify the insertion point.
3. Specify the scale.
4. Specify the rotation angle.

NOTES When inserting a drawing as a block, the base point, as defined by the *INSBASE* system variable, of the inserted drawing is the origin point of the block. The *INSBASE* system variable is set by the *BASE* command.

If a block contains attributes you will be prompted to fill out the text for each attribute in the command bar.

If the *DRAGOPEN* system variable is set to zero (OFF), drawings can be inserted as a block by dragging them from the *Windows Explorer* dialog into the current drawing; if *DRAGOPEN* = 1 (ON), the drawing will be opened instead.

The *INSUNITS* system variable defines a drawing units value for automatic scaling when inserting blocks or attaching Xrefs.


In the *Insert Block* dialog:

- The *Unit* field under *Block Unit* is set by the *INSUNITS* (Insertion Units) system variable in the source drawing or the *Block Unit* property of a block definition (see To create a block in a drawing).
- The *Factor* field expresses the relation between the value of the *INSUNITS* variable in the source drawing and the target drawing. E.g. if *INSUNITS* is Millimeters in the source and the target, the value of the *Factor* field is 0.1.
- If *INSUNITS* in the source drawing is *Unspecified*, *INSUNITSDEFSOURCE* is used instead.
- If *INSUNITS* in the target drawing is *Unspecified*, *INSUNITSDEFTARGET* is used instead.

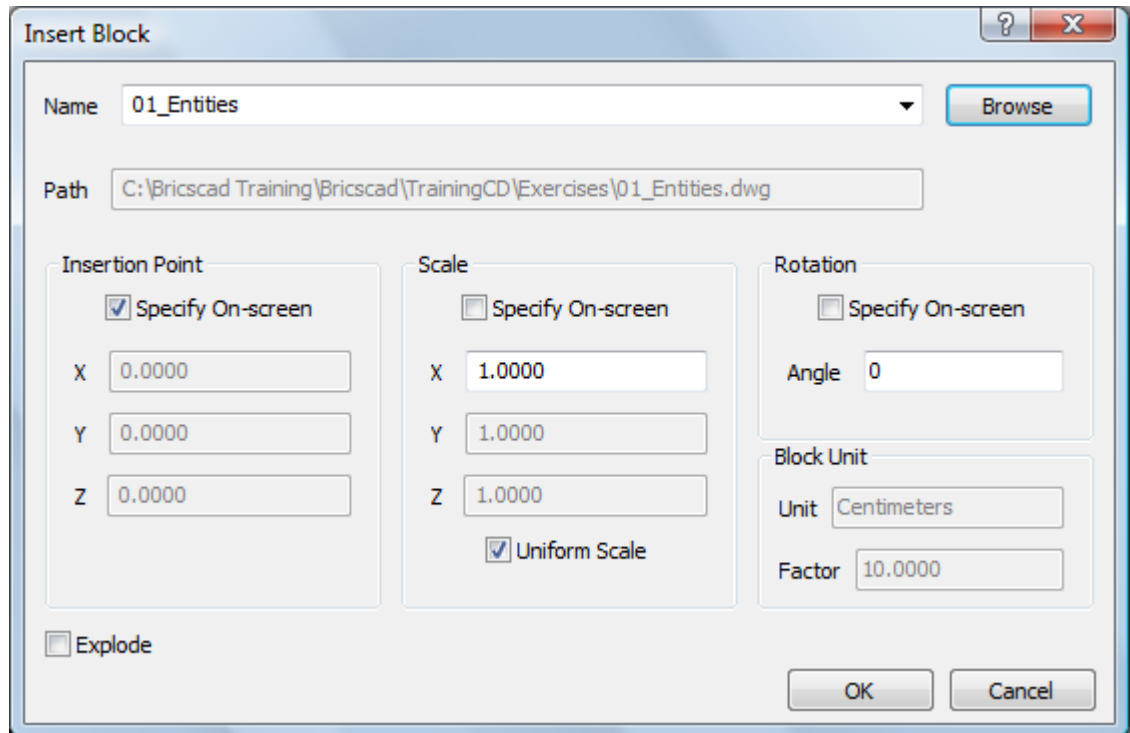
The values of *INSUNITSDEFSOURCE* and *INSUNITSDEFTARGET* are saved in the registry and therefore apply to all drawings in which *INSUNITS* is unspecified (unitless).

To insert a block

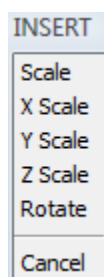
1. Do one of the following

- Click the *Insert Block...* tool button () on the *Insert* toolbar.
- Choose *Insert Block...* in the *Insert* menu.
- Type *insert* in the command bar, then press Enter.

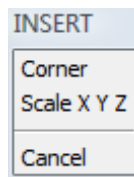
The *Inserts* dialog opens.



2. In the *Inserts* dialog, do one of the following:
 - Select an existing block definition in the *Name* list.
 - Click the *Browse* button and select a drawing file.
3. (option) Check the *Explode* option to explode the block after insertion.
4. (option) Uncheck the *Specify On-Screen* option, then specify the *Insertion Point* by keying in the coordinates in the *X*, *Y* and *Z* fields.
5. (option) Uncheck the *Specify On-Screen* option, then specify the *Scale* by keying in the scaling factors in the *X*, *Y* and *Z* fields.
Check the *Uniform Scale* option to equal the *X*, *Y* and *Z* scale factors.
6. (option) Check the *Specify On-Screen* option to specify the *Rotation* angle when inserting the block.
7. Click the *OK* button.
The command bar reads: Insertion point for block.
A prompt menu displays.



8. Specify the insertion point.
The command bar reads: Corner/XYZ/X scale factor <1.000000>:
A prompt menu displays.

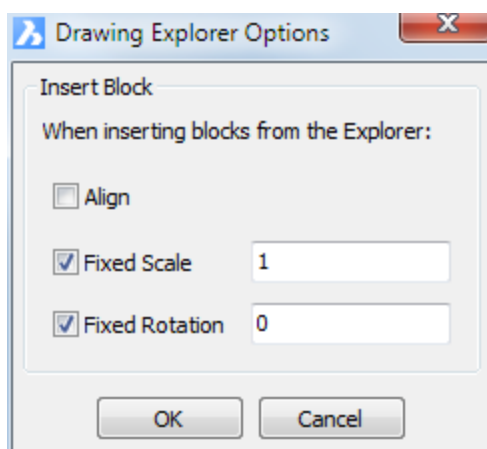


9. Right click or press Enter to accept the default *X scale factor*.
The command bar reads: Y scale factor: <Equal to X scale (1.000000)>:
10. Right click or press Enter to set the *Y scale factor* equal to the *X scale factor*.
The block is inserted.

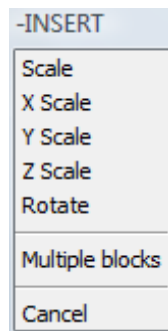
To insert a block using the Drawing Explorer

Inserting internal blocks

1. Open the Drawing Explorer - Blocks dialog.
2. (optional) Choose either *Detail View* (📐) or *Icon View* (📁).
3. (option) Edit the insert options:
 - Choose *Options...* in the *Settings* menu or select a block, then right click and choose *Options ...* in the context menu.
 - Set the insert options in the *Drawing Explorer Options* dialog box.



4. Do one of the following:
Select a block, then do one of the following:
 - Click the *Insert Block* button (📁) on the *Details* toolbar.
 - Right click and select *Insert* from the context menu
 - When in *Icon View*, double click the block.
 To select an external drawing, do one of the following:
 - Click the *Insert External* tool button (📁) on the *Details* toolbar.
 - Choose *Insert External* in the *Edit* menu.
 - Right click, then choose *Insert External* in the context menu.
 The *Drawing Explorer - Blocks* window closes.
 The command bar reads: Multiple blocks/Scale/<Insertion point for block>:
 A prompt menu displays

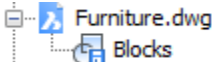


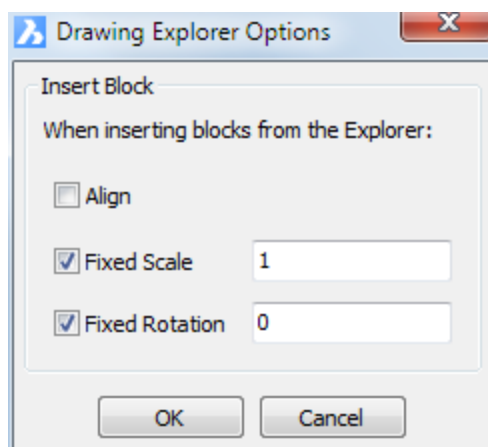
5. Follow the instructions in the command bar to insert the block.
The *Drawing Explorer - Blocks* window reopens.
6. (option) Insert more blocks.
7. Close the *Drawing Explorer - Blocks* window.

Inserting blocks from another drawing




Block Manager (Internet connection needed)

1. Open the Drawings Explorer.
 2. In the *Drawings* pane, click the *Folders* tab.
 3. (option) If not yet available in the *Local Folders* tree, add the folder of the drawing.
 4. Expand the folder.
 5. Click the *Expand* icon (+) in front of the drawing.
The *Blocks* icon of the drawing displays.
- 
6. Click the *Blocks* icon.
The blocks in the drawing display in the *Details* pane.
If necessary, click the *Icon View* button (🖼️) in the *Details* toolbar to see thumbnail images of the blocks.
 7. (option) Edit the insert options:
 - Choose *Options...* in the *Settings* menu or select a block, then right click and choose *Options ...* in the context menu.
 - Set the insert options in the *Drawing Explorer Options* dialog box.



The available options are:

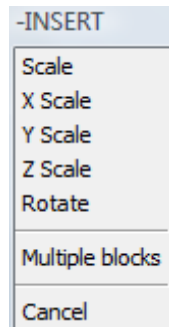
- *Align*: If checked, launches the *InsertAligned* command to insert the block.
When the *Align* option is checked, the *Fixed Scale* and *Fixed Rotation* options are not available.

- *Fixed Scale*: If checked, allows to insert the block at a fixed scale.
 - *Fixed Rotation*: If checked, allows to insert the block at a fixed rotation.
8. To insert a block, do one of the following:
 - Click the *Insert Block* button () on the *Details* toolbar.
 - When in Icon View, double click the thumbnail image of the block.
 - When in Detail view, double click the number in front of the block name.

The *Drawing Explorer* dialog closes temporarily to let you insert the block in the drawing.

The command bar reads: Multiple blocks/<Insertion point for block>:

A prompt menu displays



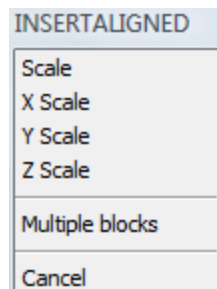
9. Follow the instructions in the command bar to insert the block.
The *Drawing Explorer* window reopens.
10. (option) Insert more blocks.
11. Close the *Drawing Explorer - Blocks* window.

To insert a block aligned with an entity

1. Make sure the appropriate Entity Snaps are active.
2. Type *insertaligned* or *insal* in the command bar, then press Enter.
The command bar reads: ? to list blocks in drawing/~ to open the file dialog/<Block to insert> <name>:
3. Do one of the following:
 - Press Enter or right click to insert the most recently placed block again.
 - Type the name of an existing block definition in the command bar, then press Enter.
 - Type the name of an external block file in the command bar, then press Enter.
 - Type ~ (tilde) then press Enter to open the file dialog.

The command bar reads: Multiple blocks/<Insertion point for block>:

A prompt menu displays.



4. Snap to an entity.
The block is aligned with the entity.

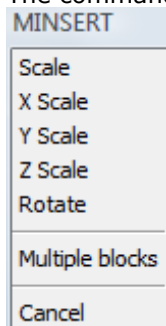
5. Click to define the insertion point.
When you move the crosshairs, the block is mirrored about the selected entity or a about a line tangent to the selected entity.
6. Click to specify the mirroring of the block.

NOTE If you type the name of block in step 3, BricsCAD will lookup the block in the block definitions in the current drawing. If the block is not found in the current drawing, the paths defined by the *SRCHPATH* system variable are searched. If the block is not found there either, BricsCAD responds 'Could not find file <blockname>'

NOTE The block is aligned as follows: The block is rotated so that its local Z-axis is parallel with the Z-axis of the active UCS, and its local X-axis should be parallel with the tangent to the entity at the position of the snap point. When you click a point on an entity the block is aligned with the entity. Then, by moving the mouse pointer around the insertion point, you can choose how the block should be mirrored. The origin point of the block lies on the entity.

To insert multiple instances of a block in a rectangular array

1. Type *minsert* in the command bar, then press Enter.
The command bar reads:
? to list blocks in drawing/~ to open the file dialog/<Block to insert>:
2. Do one of the following:
 - Type ~ (press and hold the *alt gr* key, then press the ~ key), then press Enter.
The *Insert Block* file dialog displays.
Select the drawing file you want to insert and press the *Open* button (or double click the file name).
 - Press Enter to accept the <Block to insert>.
 - Type a block name, then press Enter.
Optionally, type ? to display search for existing block definitions.
3. The block is attached to the cursor.
The command bar reads: Multiple blocks/<Insertion point for block>:



4. Specify the insertion point.
The command bar reads: Corner/XYZ/X scale factor <1.00>:
5. To define the X scale factor, do one of the following:
 - Press Enter to accept the default .
 - Type the X scale factor, then press Enter.
 - Click to define the X scale factor graphically.
 The command bar reads: Y scale factor: <Equal to X scale (current X scale)>:
6. Repeat step 5 to define the Y scale factor.
The command bar reads: Rotation angle for block <0>:

7. Do one of the following:
 - Press Enter to set the rotation angle to 0°.
 - Type a rotation angle, then press Enter.

Click to define the rotation angle graphically.
The command bar reads: Number of rows in the array <1>:
8. Type the number of rows, then press Enter or press Enter for one row.
The command bar reads: Number of columns <1>:
9. Type the number of columns, then press Enter or press Enter for one column.
The command bar reads: Vertical distance between rows, or spacing rectangle:
10. Type the row spacing distance, then press Enter.
The command bar reads: Horizontal distance between columns:
11. Type the column spacing distance, then press Enter.
The block array is created as a single entity.

NOTES If you choose *Multiple blocks* in step 3, you are prompted to create multiple instances of the block array.

It is not possible to explode a *MInsert Block* entity..

If you type the name of block in step 2, BricsCAD will lookup the block in the block definitions in the current drawing. If the block is not found in the current drawing, the paths defined by the *SRCHPATH* system variable are searched. If the block is not found there either, BricsCAD responds 'Could not find file <blockname>'

Redefining Blocks

Commands: BLOCK, -BLOCK and INSERT

To redefine a block that was created in the current drawing, you create a new block using the same name.

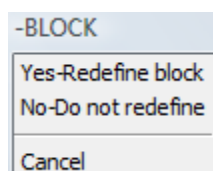
If the block was inserted from a separate drawing, reinsert that block.

When a block definition is redefined all instances of that block in the current drawing are updated automatically.

See also: Editing blocks and external references.

To redefine an internal block definition

1. Type *-block* in the command bar, then press Enter.
The command bar reads: Name for new block, or ? to list existing blocks:
2. Type the name of the block in the command bar and press Enter.
The command bar reads: The block <name> already exists. Do you want to redefine it? <N>.
A prompt menu displays.




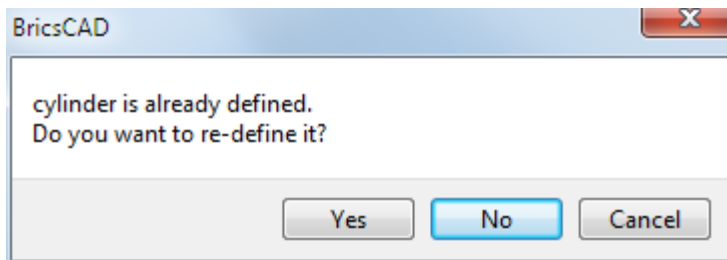
3. To redefine the block definition, do one of the following:
 - Type Y, then press Enter.
 - Choose *Yes-Redefine block* in the prompt menu.

The command bar reads: Insertion point for new block:

4. Specify the insertion point of the new block.
The command bar reads: Select entities for block.
5. Select the entities for the new block.
6. Right click to create the block.
The selected entities are removed from the drawing.
All existing instances of the block are updated.
7. (option) To bring back the selected block entities, type *oops* in the command bar, then press Enter.

To reload an external drawing as a block

1. Do one of the following
 - Click the *Insert Block...* tool button () on the *Draw* toolbar.
 - Choose *Insert Block...* in the *Draw* menu.
 - Type *insert* in the command bar, then press Enter.The *Insert Block* dialog opens.
2. On the *Insert Block* dialog click the *Browse* button, then select the external drawing file.
3. Click the *Insert* button.
An alert box displays.



4. Click the *Yes* button to redefine the existing block definition.
The command bar reads: Insertion point for block:
5. Continue with step 3 of the Inserting Blocks procedure.

Exploring Blocks

Command: EXPBLOCKS

The *Expblocks* command opens the *Blocks* section of the *Drawing Explorer* dialog box (short for "explorer blocks").

In the *Blocks Explorer* you can:

Create new blocks (📄➕)

Delete blocks (✖)

Purge unused block definitions (🗑)

Insert blocks (📄➡) in the current drawing

Insert external block files (📄📁)

Save a block as a (new) drawing (📄💾)

Cut (✂) or Copy (📄) a block definition, then Paste (📄) the block definition in another drawing

Rename a block.

Open the Blocks Explorer

6. To open the *Blocks Explorer* do one of the following:


Choose *Drawing Explorer > Blocks...* in the *Tools* menu.

Type *expblocks* in the command bar.

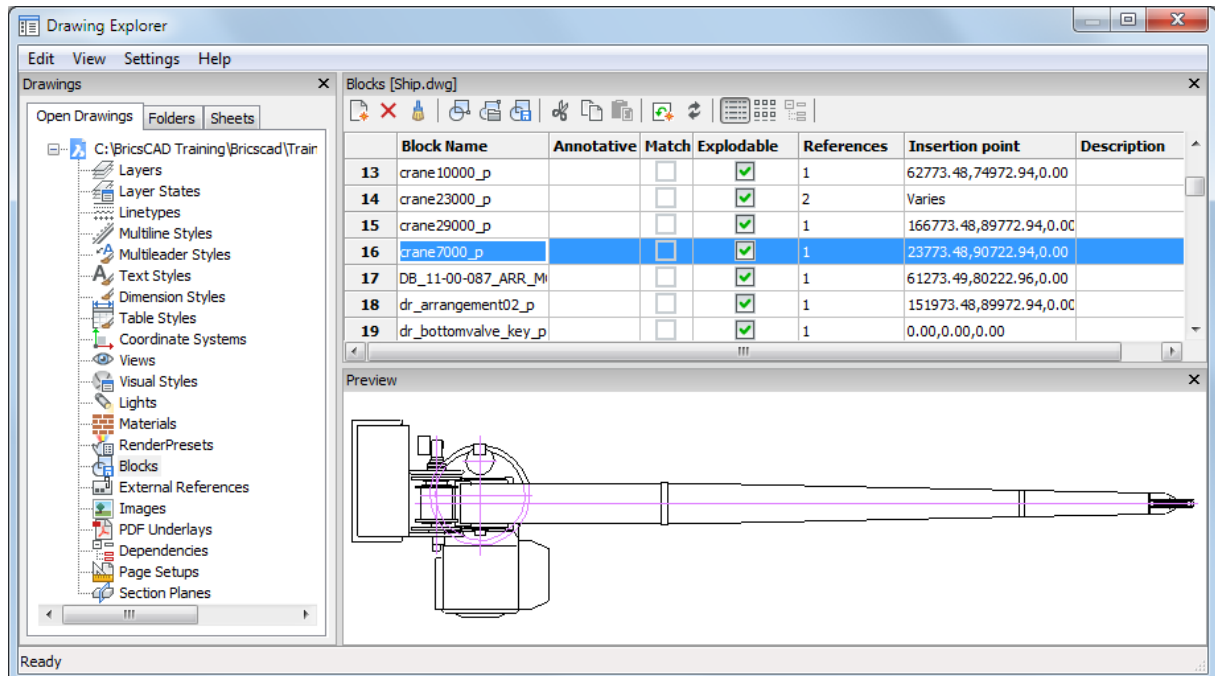
The Blocks Explorer display options

You can choose between *Detail View* (list) and *Icon View* (thumbnails) to see the blocks in the current drawing.


Open the Detail View of the Blocks Explorer

Click the *Detail View* button () in the *Drawing Explorer* toolbar. The *Detail View* button is now pressed indicating the block details are displayed.

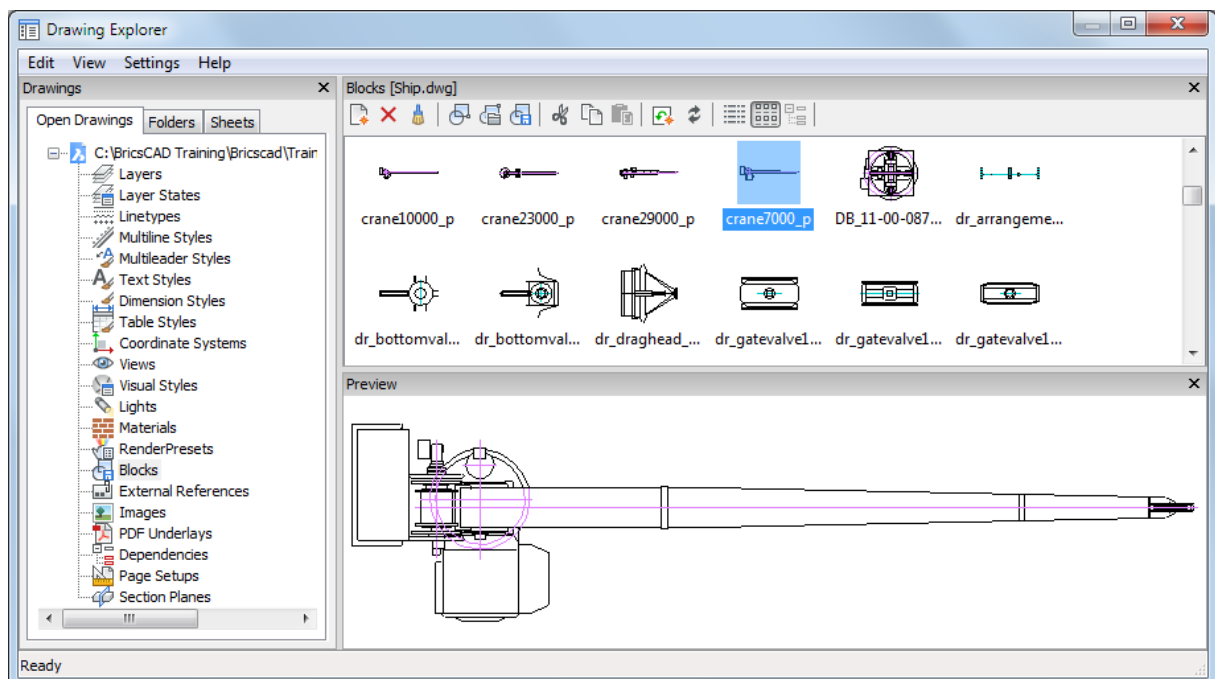
A preview of the selected block displays in the *Preview* sub-window of the *Drawing Explorer*.



Open the Icon View of the Blocks Explorer

Click the *Icon View* button () in the *Drawing Explorer* toolbar. The *Icon View* button is now pressed indicating the block icons are displayed.

A preview of the selected block displays in the *Preview* sub-window of the *Drawing Explorer*.



To rename a block


1. Open the Blocks Explorer dialog.
2. Select the block, then click the *Block Name* field.
3. Type a new name in the *Block Name* field and press Enter.

Exploding Blocks

Command: EXPLODE

If the *Explodable* property of a block is set, you can explode an inserted block to its original component entities. Exploding a block affects that single instance of the block only. The original block definition remains in the drawing and you can still insert additional copies of the original block. If you explode a block that contains attributes, the attributes are lost, but the original attribute definitions remain.

To explode a block

1. Do one of the following:
 - Click the *Explode* tool button () on the *Modify* toolbar.
 - Choose *Explode* in the *Modify* menu.
 - Type *explode* in the command bar, then press Enter.


The command bar reads: Select entities to explode.

2. Select the block(s).
3. Press Enter or right click to explode the blocks.

NOTE Exploding dissociates component entities to their next simplest level of complexity: nested blocks or polylines in a block become blocks or polylines again.

Setting the Explodable property of a block

1. Do one of the following:
 - Choose *Drawing Explorer > Blocks...* in the *Tools* menu.
 - Type *expblocks* in the command bar, then press Enter

The *Drawing Explorer - Blocks* dialog displays.
2. If necessary, click the *Detail View* tool button () in the *Drawing Explorer - Blocks* toolbar.
3. Click the checkbox in the *Explodable* field of a block to toggle the property.

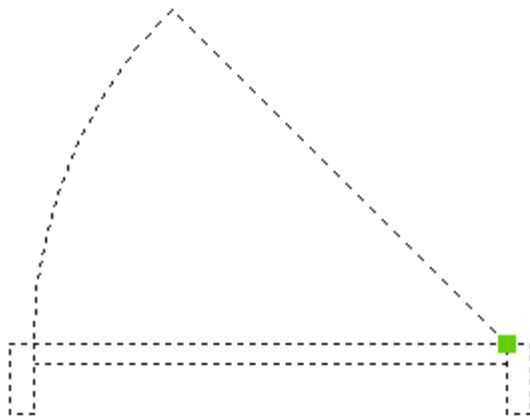
Dynamic Blocks

In V11 or higher BricsCAD partly supports dynamic blocks: you can insert dynamic blocks from your existing dynamic blocks libraries and edit the custom properties in the Properties Bar.

The creation of new dynamic blocks is not possible yet.

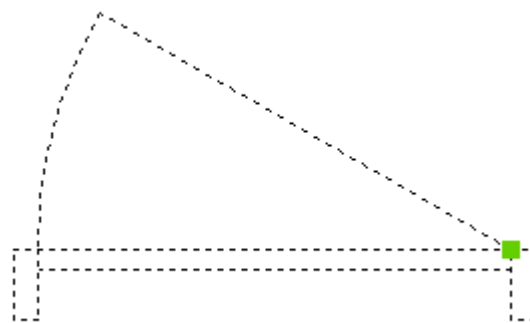
Editing dynamic blocks

1. Select the dynamic block in the drawing.
The properties of the selected block display in the Properties Bar.



Attributes	
Custom	
Door	1000
Wall	150
Opening	Right
Swing	Inside
Angle	Open 45°

2. Under *Attributes > Custom*: edit the properties needed.
The dynamic block is updated automatically.



Attributes	
Custom	
Door	1000
Wall	150
Opening	Right
Swing	Inside
Angle	Open 30°
<div> <div>Open 30°</div> <div>Open 45°</div> <div>Open 60°</div> <div>Open 90°</div> <div>Closed</div> </div>	

Working with attributes

An attribute is a particular entity that you can save as part of a block definition. Attributes consist of text-based data. You can use attributes to track such things as part numbers and prices. Attributes have either fixed or variable values. When you insert a block containing attributes, the program adds the fixed values to the drawing along with the block, and you are prompted to supply any variable values.

After you insert blocks containing attributes, you can extract the attribute information to a separate file and then use that information in a spreadsheet or database to produce a parts list or bill of materials. You can also use attribute information to track the number of times a particular block is inserted into a drawing.

Attributes can be visible or hidden. Hidden attributes are neither displayed nor plotted, but the information is still stored in the drawing and written to a file when you extract it.

General procedure to work with attributes

1. Define the attributes.

2. Create a block.

You can attach attributes to a block. Include the attributes when the program prompts you to select the entities to be included in the block definition. If a block contains attributes, the program prompts you each time you insert the block, so you can specify different values for the attributes each time you insert it into a new drawing.

3. Insert the block in a drawing.
4. (option) Edit the attributes in a block.
5. Extract attribute information.



Working with Attributes (Internet connection needed)

Defining Attributes

Command: ATTDEF and -ATTDEF

The *Attdef* command defines how a block's attributes will be created through a dialog box.


The *-Attdef* command creates block's attribute definitions at the command prompt.

General procedure to define attributes:

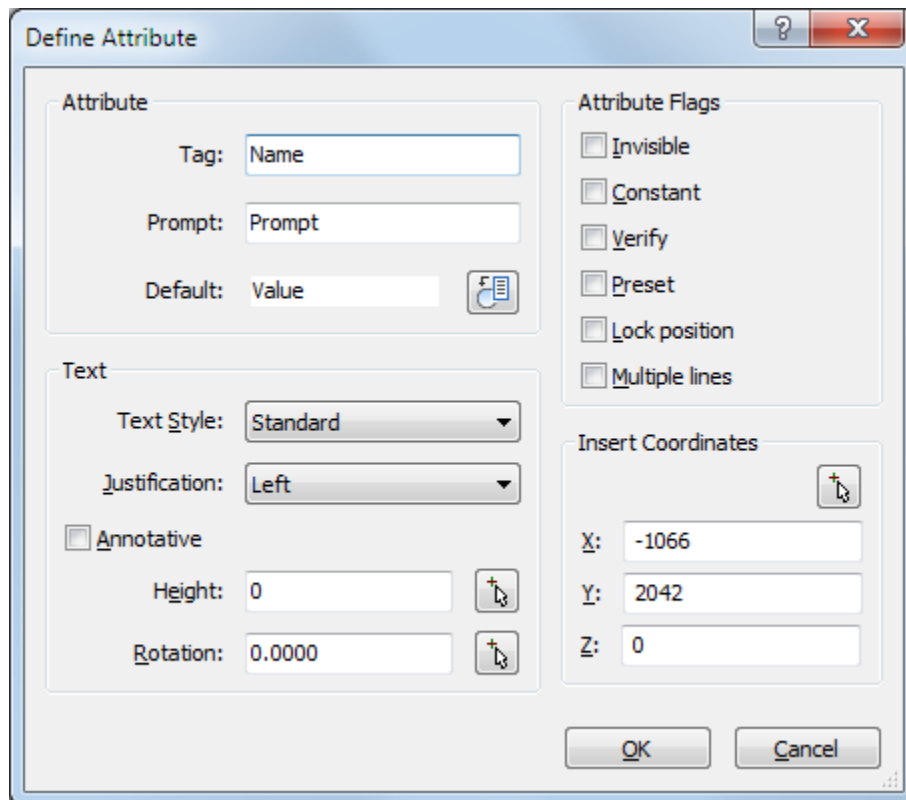
specify the characteristics of the attribute, including its name, prompt, and default value;
specify the location
define the text formatting
optionally: set the attribute flags (hidden, fixed, validate or predefined).

To define an attribute

1. Do one of the following:

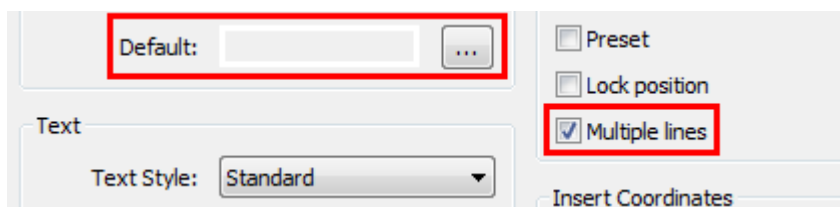
- Click the *Define Attributes...* tool button () on the *Tools* toolbar.
- Choose *Define Attributes...* in the *Tools* menu.
- Type *attdef* in the command bar, then press Enter.

The *Define Attribute* dialog opens.



2. Define the Attribute:

- Type a name in the *Tag* field.
The *Tag* identifies each occurrence of an attribute in the drawing. The name you type in the *Tag* field can be any combination of characters except spaces. Lowercase letters are automatically converted to uppercase.
- Type a prompt in the *Prompt* field.
The attribute prompt displays when you insert a block containing the attribute. If a prompt is omitted, the *Tag* is used as the prompt instead.
- Do one of the following:
- Type a default value in the *Default text* field.
For variable attributes, the default value is replaced by the actual value when you insert a block containing the attribute.
- Click the *Insert Field* button (📄) to assign a field value to the attribute (see Using sheet set and sheet properties in a title block).
- If the *Multiple lines* flag is checked, the *Default text* field is dimmed.



Click the button next to the *Default text* field.

The *Define Attribute* dialog temporarily closes to let you create a Mtext entity.

- Do one of the following:
- Specify the x-, y-, and z-coordinates for the attribute insertion point.
- Click the *Pick Point* button (📍) to specify the insertion point in the drawing.
The command bar reads: Select insertion point.
The *Define Attribute* dialog closes temporarily.
- (option) Set the *Invisible* attribute flag.
Attributes with the *Invisible* flag set do not display nor print.

If the *ATTMODE* (Attribute Display Mode) variable is set to 2, all attributes display, including *Hidden* attributes.

- (option) Set the *Constant* attribute flag.
Attributes with the *Constant* flag set cannot be edited.
 - (option) Set the *Verify* attribute flag.
Attributes with the *Verify* flag set must be explicitly validated when a block containing such attributes is inserted.
 - (option) Set the *Preset* attribute flag.
When inserting a block you are not prompted to define attributes of which the *Preset* flag is set. You can edit attribute afterwards though.
 - (option) Set the *Lock Position* attribute flag to lock the position of the attribute within the block reference.
Unlocked attributes can be moved relative to the rest of the block using grip editing and multiline attributes can be resized.
 - Define the *Text* properties: *Text Style*, *Justification*, *Annotative*, *Text height* and *Rotation*.
3. Click the *OK* button to create the attribute in the drawing.
The *Define Attribute* dialog closes.
 4. (option) Right click to reopen the *Define Attribute* dialog, then repeat steps 2 and 3 to create another attribute.

Using sheet set properties in a title block

1. Create the sheet set.
2. Define the custom sheet set and sheet properties.
3. Open a sheet (layout) from the sheet set.
4. Create the title block source entities in the paper space layout:
 - Create the borders, line work and fixed texts.
 - Create attributes with fields that refer to sheet set, subset and sheet properties (see To define an attribute).

See the SHEETSET command to learn more about default and custom sheet set, subset and sheet properties.

Custom sheet set properties:

Sheet Set Custom Properties	
Architect	Bricsys & Partners
ArchitectAddress	Bellevue 5/201
ArchitectCity	9050 Gent
ArchitectPhone	09 244 01 91
Contractor	Robert Stone Ltd.
ContractorAddress	123 Main Street
ContractorCity	New Town
Owner	Carpenter
OwnerAddress	1 Church Road
OwnerCity	New Town
OwnerTitle	Mr. & Mrs.
SiteAddress	48 Brick Lane
SiteCity	New Town
SiteReference	Section A no. 123

Sheet properties and sheet custom properties:

Sheet	
Number	08
Title	Elevation South
Description	South elevation drawing
Publish	<input checked="" type="checkbox"/> Include for Plot/Publish
Layout	LAYOUT1 (C:\BricsCAD Trainin
Revision Number	1
Revision Date	15/08/2013
Issue Purpose	Final Design
Category	Elevation
Sheet Custom Properties	
Date	12/08/2013
PaperSize	A2
Printer	pdf
Remark	none
Scale	1/50

5. Create a block from the title block source entities.
6. Copy the block to drawing in one of your block folders (see To insert a block form another drawing).
7. Delete the title block source entities.
8. The title block can be inserted in the other sheets of the sheet set and in any other sheet set where the same custom sheet set and sheet properties exist. All fields that refer to default or custom sheet set, subset or sheet properties are filled out automatically. If no value is available, a field displays hyphens (----) (see also Placing blocks with attributes).

Editing attribute definitions

Command: DDEDIT

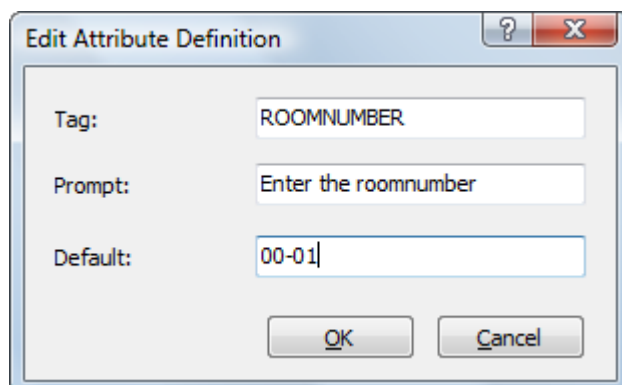
The *Ddedit* command edits single-line text, multi-line text, attribute definitions, and attribute text (short for "dynamic dialog editor").

You can edit an attribute definition only before it is saved as part of a block definition.

To edit tag, prompt or default text of an attribute

1. Do one of the following:
 - Type *ddedit* in the command bar, then press Enter and select the attribute.
 - Double click the attribute.

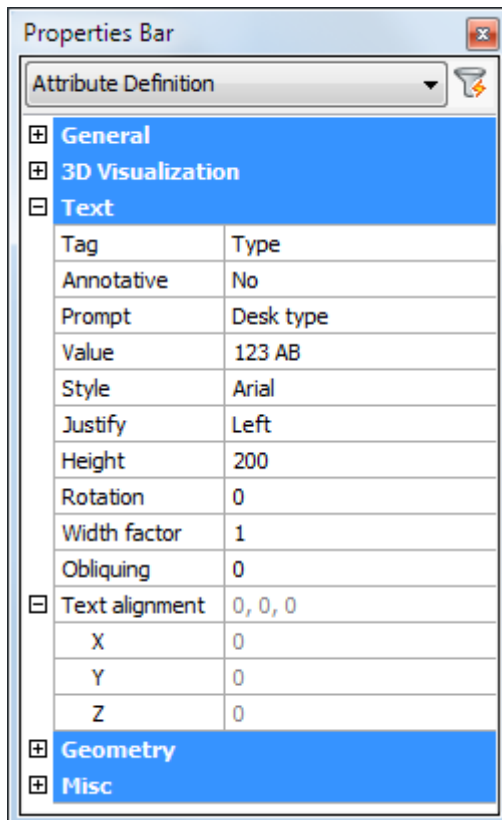
The *Edit attribute definition* dialog opens.



2. (option) Type a new name in the *Tag* field.
Lowercase letters are converted to uppercase automatically.
3. (option) Type a new prompt in the *Prompt* field.
If you leave the *Prompt* field empty, the attribute *Tag* will be used as the prompt instead.
4. (option) Type a new text in the *Default* field.
5. Press the *OK* button to save your changes.

To edit an attribute definition

1. Click the attribute.
The attribute properties display in the *Properties Bar*.



The screenshot shows the 'Properties Bar' dialog box with the 'Attribute Definition' tab selected. The 'Text' section is expanded, showing a table of properties for the attribute.

Tag	Type
Annotative	No
Prompt	Desk type
Value	123 AB
Style	Arial
Justify	Left
Height	200
Rotation	0
Width factor	1
Obliquing	0
Text alignment	0, 0, 0
X	0
Y	0
Z	0

2. Modify the attribute properties in the *Properties Bar* fields.
3. Press the Esc key to stop editing.

NOTE Make sure the *Properties Bar* is open before starting this procedure.

Placing blocks with attributes

Commands: INSERT and -INSERT

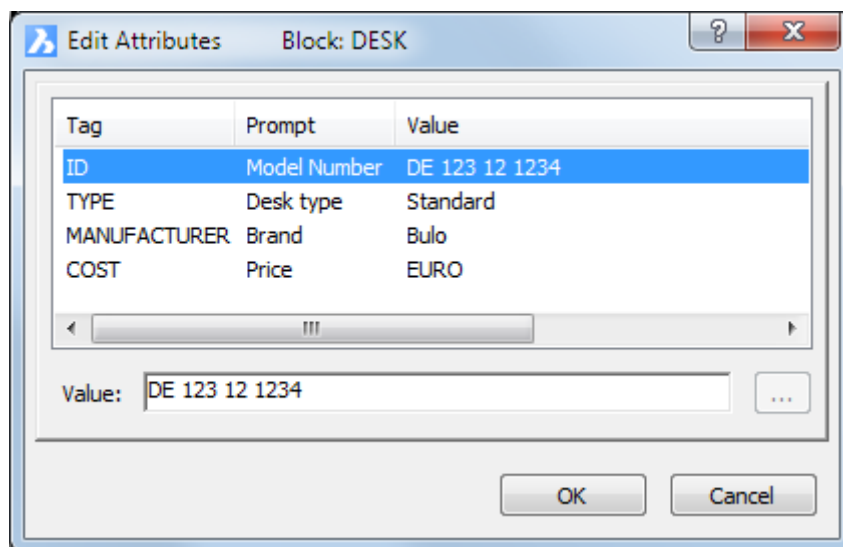
When you place a block which contains attributes, you will be prompted to define the attributes in the command bar or through a dialog box.

If the ATTDIA system variable is ON, attributes display in a dialog box.

If the ATTDIA system variable is OFF, you are prompted in the command bar to define the attributes.

To place a block with attributes

1. Launch the Insert Block... tool.
2. Position the block in the drawing.
All block entities are placed in the drawing, except the attributes.
The command bar reads: <Attribute Prompt> <Default Text>:
or
A dialog box displays:



3. Do one of the following:
 - In the command bar: The command bar reads: <Attribute Prompt> <Default Text>:
 - Press Enter or right click to accept the default text.
 - Type a new text in the command bar, then press Enter and define the next attribute.
 - In the *Edit Attributes* dialog box:
 - Select an attribute in the list.
The current value displays in the *Value* field.
 - Type a new value in the *Value* field.
 - Press *OK* when all attributes are properly defined.


Editing attributes attached to blocks

Command: EATTEDIT

The *Eattedit* command edits the attributes in blocks. You are prompted to select a block with editable attributes. When you select such block, the *Attribute Editor* dialog displays. If the selected block does not contain attributes or if the selected entity is not a block you are prompted to select another block.

Using the Attribute Editor

1. Do one of the following

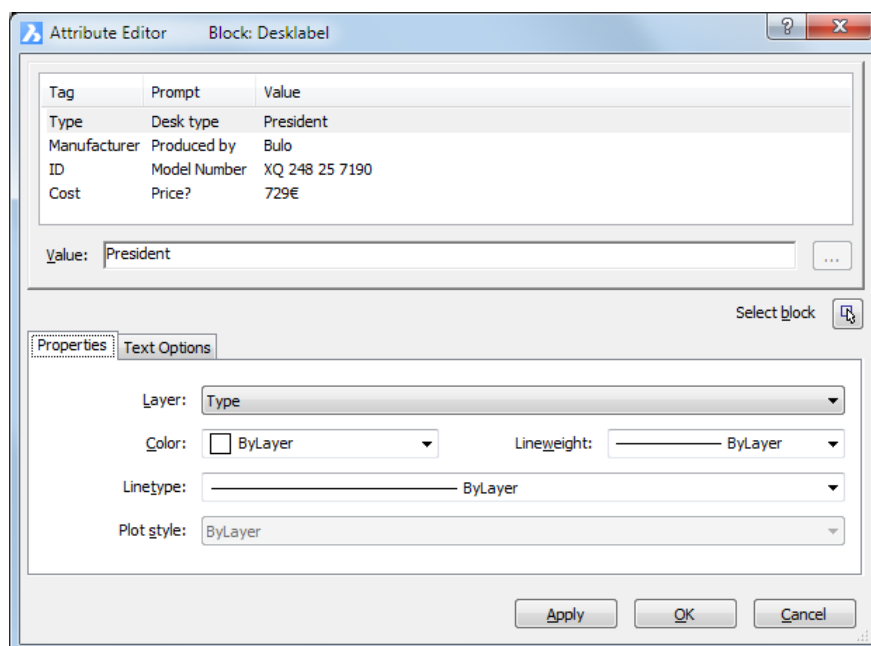
- Click the *Edit Block Attributes...* tool button () on the *Attributes* toolbar.
- Choose *Edit Block Attributes...* in the *Tools | Attributes* menu.
- Type *eattedit* in the command bar, then press Enter.

The command bar reads: Select a block:

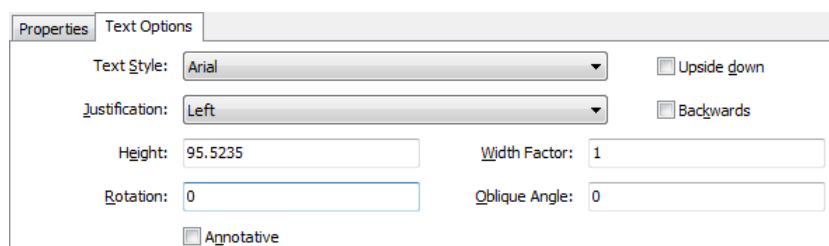
or


Double click a block containing attributes, then go to step 3.

2. Select a block with attributes.
The *Attribute Editor* dialog opens.



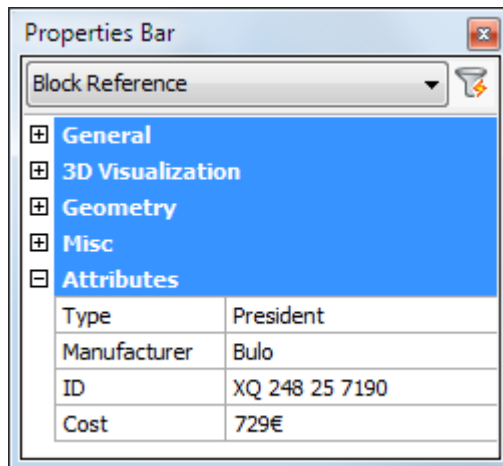
3. Select an attribute in the list.
The current value of the selected attribute displays in the *Value* field.
4. (option) Type a new value in the *Value* field.
5. (option) Click the *Properties* tab to modify *Layer*, *Color*, *Lineweight* and/or *Linetype*.
6. (option) Click the *Text Options* tab to edit the text properties.



7. (option) Click the *Apply* button to apply the changes.
8. (option) Repeat steps 3 through 6 to edit another attribute.
9. Do one of the following:
 - Click the *OK* button to close the *Attribute Editor* dialog and apply the changes.
 - Click the *Apply* button to apply the changes and keep the *Attribute Editor* dialog open.
 - Click the *Cancel* button to close the *Attribute Editor* dialog. Changes that are not applied yet are not saved.
 - Click the *Select* button () to select another block.
The *Attribute Editor* dialog temporarily closes to let you select another block in the drawing.

Editing attributes in the Properties bar

1. Select a block containing attributes.
The current value of the attributes display in the Properties bar.
2. Click the attribute you want to edit.
The settings field of the selected attribute is active.



3. Type a new value, then press enter or select another attribute.

Extracting attribute information

Command: ATTEXT and - ATTEXT

The *Attext* command copies data from attributes to a text file via a dialog box.

The *-Attext* command copies data from attributes to a text file via the command line.

The text file can be imported in a database or a spreadsheet.

You can save the file in any of the following formats:

- *Comma Delimited Format* (CDF): Contains one line for each instance of a block, with individual attribute fields separated by commas. Character string fields are enclosed with single quotation marks. You must specify a template file when extracting to a CDF file.
- *Space Delimited Format* (SDF): Contains one line for each instance of a block. Each attribute field has a fixed length; there are no separators or character string delimiters. You must specify a template file when extracting to a SDF file.
- *Drawing Exchange Format* (DXF): Creates a subset of a standard DXF file (a *.dxx file) containing all the information about each block, including the insertion points, rotation angles, and attribute values. No template file is required.

Before extracting attributes to a CDF or SDF file, you must create a template file. The template file is an ASCII text file that specifies the attribute data fields to be written in the extract file. Each line of the template file specifies one attribute field. BricsCAD recognizes 15 different fields, which contain elements such as the block name, the x-, y-, and z-coordinates of its insertion point, the layer on which it is inserted, etc. You can include any of these fields. The template file must include at least one attribute name.

Each line in the template file must start with the field name. Block name and insertion-point values must begin with BL:. The next nonblank character must be either a C (indicating a character string field) or an N (indicating a numeric field). This character is then followed by three digits indicating the width of the field (in characters). The final three digits indicate the number of decimal places (for numeric fields). In the case of character fields, the last three digits must be zeros (000). A typical template file is similar to the one shown here:

A		B	
1	BL:DESK	4	C012000
2	BL:X	5	N007001
	BL:Y	6	N007001
	TYPE		C012000
3	MANUFACTURER		C012000
	ID		C012000
	COST		N008002

A. Field name

B. Field format

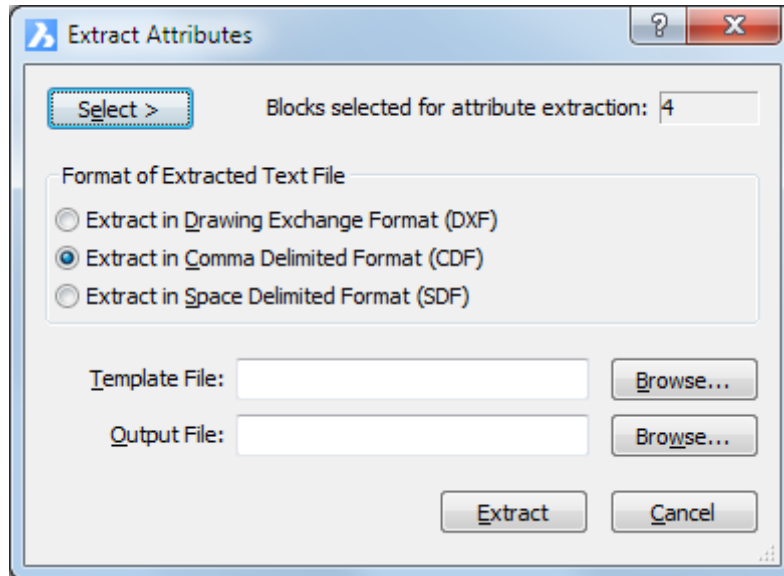
1. Block name (must be preceded by BL:)
2. Coordinate fields (must be preceded by BL:)
3. Attribute tags
4. Indicates the field data type: C for character fields, N for numeric fields
5. Field width (number of characters)
6. Number of decimal places for numeric fields, 000 for character fields

To extract attribute information

1. Do one of the following:

- Click the *Extract Attributes* tool button on the *Tools* toolbar.
- Choose *Extract Attributes* in the *tools* menu.
- Type *atttext* in the command bar, then press Enter.

The *Extract Attributes* dialog opens.



2. Click the *Select* button.
The *Extract Attributes* dialog closes temporarily.
3. Select the blocks, then press Enter or right click.
The *Extract Attributes* dialog reopens.
4. Select the text file format: *DXF* (drawing exchange format), *CDF* (comma delimited format) or *SDF* (space delimited format).
5. Select the *Template file*.
6. Specify the *Output file* name.
7. Click the *Extract* button.
The command bar reads: <number> records extracted.

Working with external references

You can link entire drawings to the current drawing as external references. Unlike inserting a drawing as a block, in which you add all the entities from the separate drawing into the current drawing, external references attach a pointer to the external file. The entities in the external reference appear in the current drawing, but the entities themselves are not added to the drawing. Thus, attaching an external reference does not significantly increase the size of the current drawing file.

External references provide additional capabilities not available when you insert a drawing as a block. When you insert a drawing as a block, the entities are stored in the drawing. Any changes you make to the original drawing are not reflected in the drawing in which you inserted it. When you attach an external reference, however, any changes you make to the original drawing file are reflected in the drawings that reference it. These changes appear automatically each time you open the drawing containing the external reference. If you know that the original drawing was modified, you can reload the external reference anytime you're working on the drawing.

External references are useful for assembling master drawings from component drawings. Use external references to coordinate your work with others in a group. External references help reduce drawing file size and ensure that you are always working with the most recent version of a drawing.

Commands: XREF, -XREF, XATTACH and XCLIP

The *Xref* command attaches DWG files to the current drawing through the Drawing Explorer (short for "external reference").

The *-Xref* command attaches DWG files to the current drawing through the command bar.

The *Xattach* command attaches externally-referenced drawings through a dialog box (short for "external attach").

The *Xclip* command clips externally-referenced drawings, and adjust front and rear clipping planes (short for "xref clip").



Attaching an external reference

Command: XATTACH

The *Xattach* command attaches a drawing as an external reference (xref). The attached xref appears as an insert in the current drawing. Like blocks, attached xrefs can be nested. The attached drawing is based on the most recently saved version.

The origin point used when attaching an external reference is controlled by the *INSBASE* (insertion base point) settings variable. Use the *BASE* command to define the *INSBASE* variable in the external reference drawing.

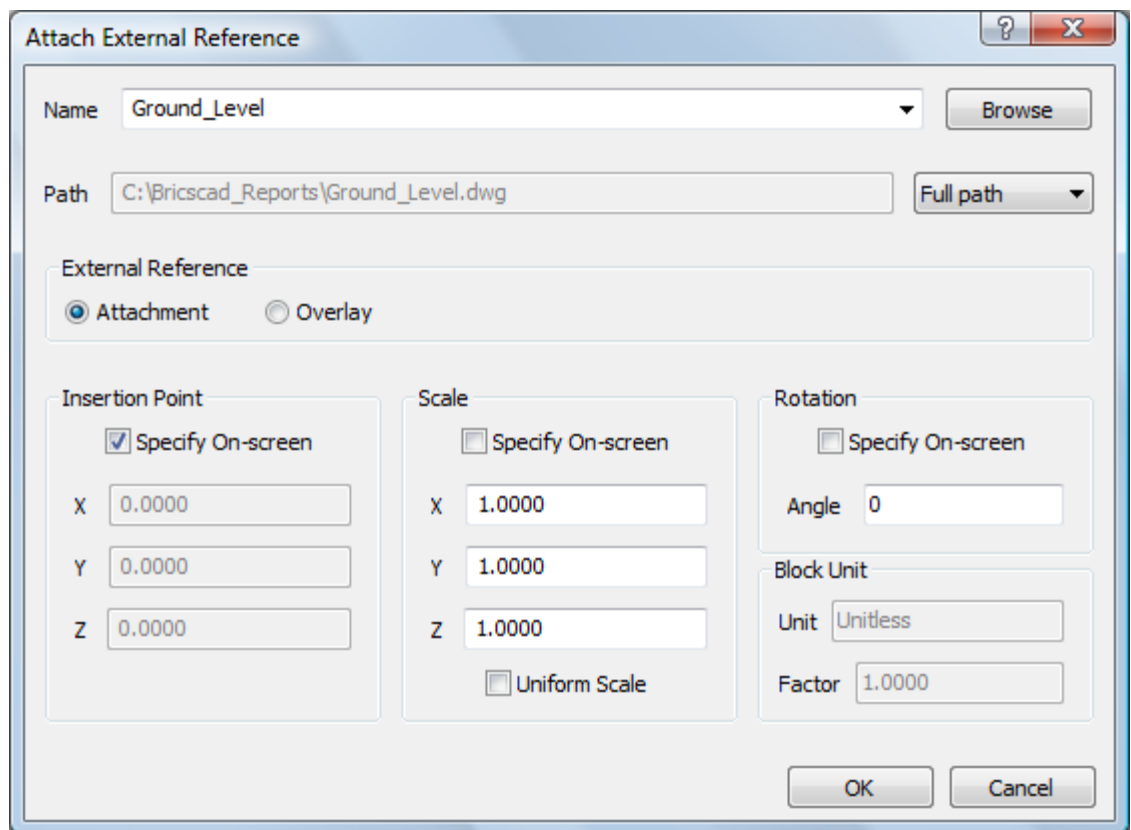
To attach an external reference

1. Do one of the following
 - Click the *Attach Xref...* tool button () on the *Insert* toolbar.
 - Choose *Attach Xref...* in the *Insert* menu.
 - Type *xattach* in the command bar, then press Enter.
 - Click the *Attach Xref* tool button () on the *Details* toolbar on the Drawing Explorer / Xrefs window.

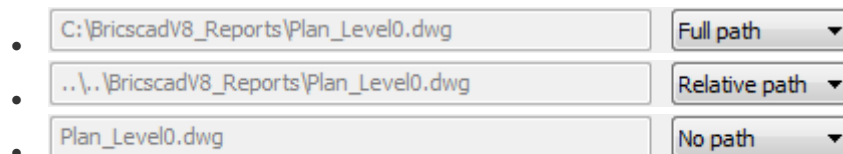
The *Attach External Reference* dialog window displays.

2. Select the drawing you want to attach, then press the *Open* button on the *Open* dialog window.

The *Attach External Reference* dialog window opens:



3. The full path to the selected drawing displays in the *Path* field.
(option) Select the *Path Type*:



4. Choose the *External Reference Type*:
 - *Attachment*: If the current drawing is attached as an xref to another drawing, an attachment becomes a nested xref.
 - *Overlay*: An overlay is ignored when the drawing to which it is attached is then attached as an xref to another drawing.
5. Choose whether to specify the *Insertion Point* on screen or in the X-, Y-, Z-fields.
6. Choose whether to specify the *Scale* on screen or in the X-, Y-, Z-fields.
Check the *Uniform Scale* option to specify the scale in the X-field.
7. Choose whether to specify the *Rotation* angle on screen or in the *Angle* field.
8. Click the *OK* button to attach the Xref.
Depending on the options in steps 5, 6 and 7 you are prompted to specify the insertion point, scale and/or rotation angle.

NOTE The *Unit* field under *Block Unit* is set by the *INSUNIT* (Insertion Units) system variable in the external reference. The *Factor* field expresses the relation between the value of the *INSUNIT* system variable in the xref and the parent drawing.

Managing Xrefs

Command: XREF

The *Xref* command opens the *External References* section of the *Drawing Explorer* dialog box.

	Reference Name	Loaded	Size	References	Type	Date	Saved Path	Found Path
1	ELEVATION_EAST		99.0 Kb	1	Attach	17- 5-2013	C:\Projects\Template (me	C:\Projects\Barcelona\Drawing\Elevation_East.dwg
2	ELEVATION_NORTH		105.1 Kb	1	Attach	17- 5-2013	C:\Projects\Starter Templ	C:\Projects\Barcelona\Drawing\Elevation_North.dwg
3	ELEVATION_SOUTH		76.3 Kb	1	Attach	17- 5-2013	C:\Projects\Starter Templ	C:\Projects\Barcelona\Drawing\Elevation_South.dwg
4	ELEVATION_WEST		37.3 Kb	1	Attach	17- 5-2013	C:\Projects\Starter Templ	C:\Projects\Barcelona\Drawing\Elevation_West.dwg
5	ISOMETRIC_NE		63.2 Kb	1	Attach	17- 5-2013	C:\Projects\Starter Templ	C:\Projects\Barcelona\Drawing\Isometric_NE.dwg
6	ISOMETRIC_SW		68.6 Kb	1	Attach	17- 5-2013	C:\Projects\Starter Templ	C:\Projects\Barcelona\Drawing\Isometric_SW.dwg
7	PLAN_LEVEL 0		61.4 Kb	1	Attach	17- 5-2013	C:\Projects\Starter Templ	C:\Projects\Barcelona\Drawing\Plan_Level 0.dwg
8	SECTION_AB		90.6 Kb	1	Attach	17- 5-2013	C:\Projects\Starter Templ	C:\Projects\Barcelona\Drawing\Section_AB.dwg
9	SECTION_CD		51.3 Kb	1	Attach	17- 5-2013	C:\Projects\Starter Templ	C:\Projects\Barcelona\Drawing\Section_CD.dwg

The *Drawing Explorer - External References* dialog allows to:

- Attach a drawing as an external reference ().
- Detach an external reference ().
- Reload an external reference ().
- Unload an external reference ().
- Bind an external reference ().
- Insert an external reference ().
- Modify the attachment type: *Attach* or *Overlay*.
- Modify the *Saved Path* of an xref.
- Modify the *Found Path* of an overlay.

To detach external references

1. Select the xref(s) you want to detach.
Press and hold the Ctrl key to select multiple xrefs or to unselect a selected xref.
2. Click the *Detach Xref* tool button () on the *Details* toolbar.
The selected xref(s) are detached. You are not prompted to confirm the detachment.

To unload external references

Do one of the following:

- Click the *Loaded* column of a loaded Xref.
- Select the Xref(s), then either click the *Unload Xref* tool button () on the *Details* toolbar or click the *Loaded* column of one of the selected Xrefs.

To reload external references

Do one of the following:

- Click the *Loaded* column of an unloaded Xref.
- Select the Xref(s), then either click the *Reload Xref* tool button () on the *Details* toolbar or click the *Loaded* column of one of the selected Xrefs.

Editing blocks and external references

Commands: REFEDIT, -REFEDIT, REFSET and REFCLOSE

The *Refedit* command edits block references and externally-referenced drawings (Xrefs) through a dialog box (short for "reference editor"). *Refedit* is referred to as *in-place reference editing*. *RefEdit* either works on a portion of the reference or on the entire drawing or all entities in the block.

The *-Refedit* command edits block references and externally-referenced drawings through the command bar.

The *Refset* command adds and removes entities from the reference being edited.

The *Refclose* command closes the reference editor.


The *XFADECTL* system variable sets the fading level for the entities which are not in the working in the current refedit session.



Reference Editing (Internet connection needed)

To edit Xrefs or blocks

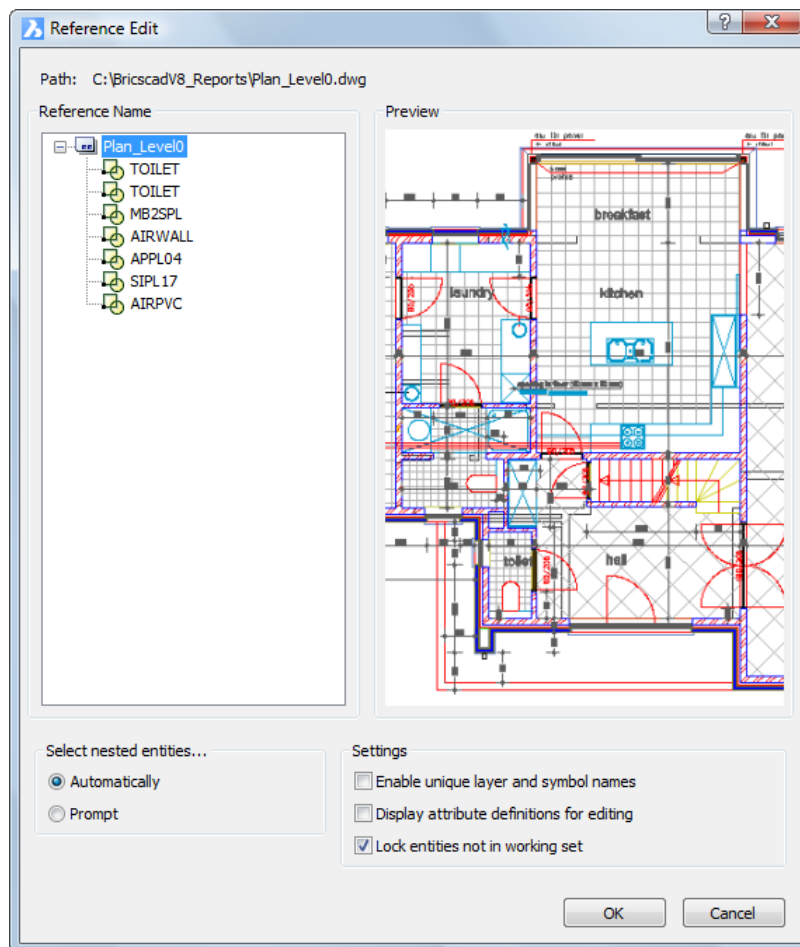
1. Do one of the following:

- Click the *Refedit* tool button () on the *Refedit* toolbar.
- Type *refedit* in the command bar, then press Enter.

The command bar reads: Select reference:
Click the Xref or block.





- Double click the Xref or block.

The *Reference Edit* dialog opens.



2. The *Reference Edit* dialog helps you to visually identify the reference to edit and controls how the reference is selected.
 - **Path:** Location of the selected reference; if the reference is a block, no path is displayed.
 - **Reference Name:** The name of the currently selected reference and any references, that are nested within the selected reference.
 - **Preview:** Displays the reference as it was last saved in the drawing. Please note that the preview image is not updated when changes are saved back to the reference.
3. (option) If multiple nested references display in the *Reference name* tree, select a reference to modify.
Click the expand/collapse button to show/hide nested references.
You can edit only one reference a time.
4. Select one of the following to *Select nested entities ...*:
 - **Automatically:** All the entities and nested blocks in the selected reference are included in the reference editing session.
 - **Prompt:** After you close the *Reference Edit* dialog, you are prompted to select the entities and nested blocks in the reference that you want to edit. Only these entities will be temporarily extracted and be made available for modification within the context of the current drawing.

This temporary working set of entities can be edited and then saved back to update the reference being edited. To distinguish 'current drawing' from 'editing set' BricsCAD displays all the entities of the current drawing as faded. The fading level is controlled through the *XFADECTL* settings variable, of which the default value is set to 50.

5. Check the *Settings*:
 - *Enable unique layer and symbol names*:
 - If selected, named objects in xrefs are altered, adding a \$\$\$ prefix.
 - If cleared, the names of layers and other named objects remain the same as in the reference drawing.
 - *Display attribute definitions for editing*: Controls whether attribute definitions in blocks are extracted and displayed during reference editing.
 - *Lock entities not in working set*: Entities in the reference being edited that are not included in the temporary working set (see step 5) cannot be manipulated, nor entities in the host drawing or other Xrefs. This behavior is similar to objects on locked layers.
6. Click the *OK* button to start editing the working set (see step 5). The *Reference Edit* dialog closes.
7. (option) Click the *Add to Refedit* tool button () to add entities to the working set. The selected entities will be removed from the host drawing and added to the reference being edited when the working set is saved back.
8. (option) Click the *Remove from Refedit* tool button () to remove entities from the working set. The selected entities will be removed from the reference being edited when the working set is saved back. These entities are added to the host drawing.
9. Do one of the following:
 - Click the *Refclose and Save* button () to save the changes back to the Xref source drawing or the block definition in the current drawing.
 - Click the *Refclose and discard* button () to stop the refedit session without saving. The source drawing or block definition is left unchanged.

Working with groups

Commands: GROUP and -GROUP


The *Group* command creates and modifies named groups of entities, in a dialog box.

The *-Group* command creates and modifies named groups of entities, at the command line.

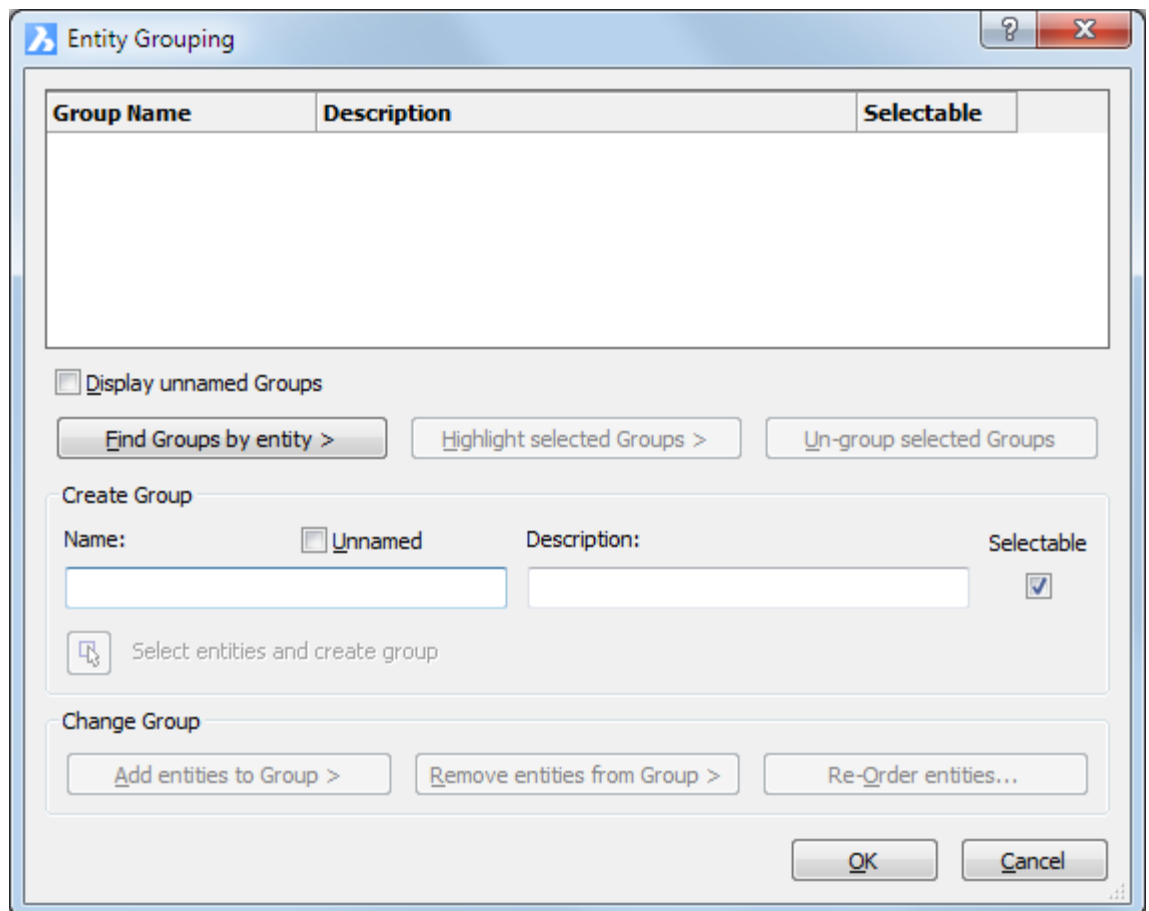
Working with groups is a technique to manipulate a set of several objects as though it were a single object. A group can also be considered as a 'Named Selection'. You can pick a single object and the entire set is selected as though it were a block, or you can edit individual objects without affecting the rest of the group and without breaking up the group. You can use nested groups if necessary and a single entity can be a member of different groups.

NOTE Make sure the value of the *PICKSTYLE* variable is set to either 1 or 3 to enable the selection of groups.

To create a group


1. Do one of the following:
 - Click the *Group* tool button () on the *Tools* toolbar.
 - Choose *Group* in the *Tools* menu.
 - Type *group* in the command bar, then press Enter.

The *Entity Grouping* dialog window opens:




2. Do one of the following:
 - Type a name in the *Name* field.
 - Check the *Unnamed* option to create an unnamed group.
3. (option) Type a description in the *Description* field.
4. (option) Check / uncheck the *Selectable* option.
5. Click the *Select entities and create group* button.
The *Entity Grouping* dialog temporarily closes to let you select entities.
6. Select the entities you want to include in the new group.
7. Right click to conclude the selection of entities.
The *Entity Grouping* dialog reopens.
The newly created group is added.
8. Click the *OK* button to close the *Entity Grouping* dialog

To modify a group

1. Do one of the following:
 - Click the *Group* tool button () on the *Tools* toolbar.
 - Choose *Group* in the *Tools* menu.
 - Type *group* in the command bar, then press Enter.

The *Entity Grouping* dialog window opens.
2. Select the group you want to modify.
3. (option) Double click in the *Name* field to edit the group name.
4. (option) Double click in the *Description* field to edit the group description.
5. (option) Click in the *Selectable* column to toggle the *Selectable* option on/off.
6. (option) Click the *Add entities to group* button to add new entities to the group.
The *Entity Grouping* dialog temporarily closes to let you select the new group entities.
7. (option) Click the *Remove entities from group* button to remove entities from the group.
The *Entity Grouping* dialog temporarily closes to let you select the entities.
8. Click the *OK* button to close the *Entity Grouping* dialog


To ungroup entities

1. Do one of the following:
 - Click the *Group* tool button () on the *Tools* toolbar.
 - Choose *Group* in the *Tools* menu.
 - Type *group* in the command bar, then press Enter.

The *Entity Grouping* dialog window opens.
2. Select the group or groups you want to delete.
3. Click the *Un-group selected groups* button.
The selected groups are removed.

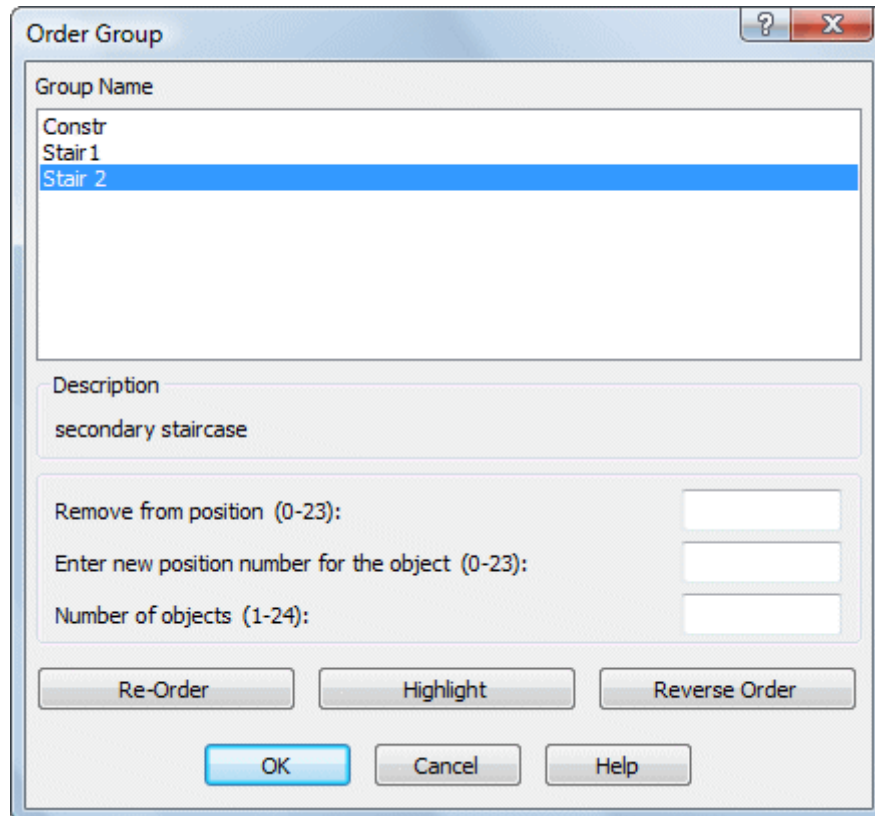
NOTE When you ungroup entities, the entities remain in the drawing but the group is deleted from the drawing.

To change the order of entities

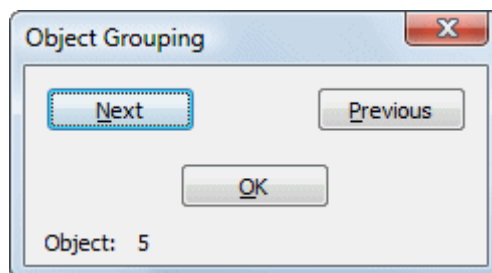
1. Do one of the following:
 - Click the *Group* tool button () on the *Tools* toolbar.
 - Choose *Group* in the *Tools* menu.
 - Type *group* in the command bar, then press Enter.

The *Entity Grouping* dialog window opens.

2. Click the *Re-Order entities* button.
The *Order Group* dialog opens:

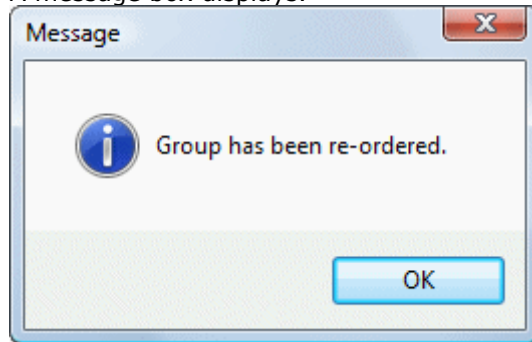


3. (option) Click the *Highlight* button to see the current position of each entity.
The *Object Grouping* box opens.



4. (option) Click the *Reverse Order* button to reverse the order of all entities in the group.

5. (option) To change the order of a single entity do the following:
 - In the *Remove from position* field type the current position of the entity you want to move.
 - In the *Enter new position number for the object* field type the new position.
 - (option) Type the number of objects you want to reorder in the *Number of objects* field.
6. Click the *Re-Order* button.
A message box displays.



Working with underlays

Commands: PDFATTACH, PDFLAYERS, PDFCLIP

PDF files can be attached as an underlay to a drawing file. Much like raster image files and external references (Xrefs) a PDF underlay is not part of the drawing, but is linked to it. In the parent drawing the path to the underlay is saved. You can edit the path to make sure the correct PDF is found. By default the folder of the parent drawing is searched first. If the PDF underlay is not found there, the saved path folder is searched. If the PDF underlay is not found there either, BricsCAD reports a 'Missing or invalid reference' in the drawing.

You can snap to the geometry in the PDF underlay if the PDFOSNAP system variable is On. Please notice that when snapping to geometry in a PDF underlay the precision is far less than snapping to entities in the drawing or an xref.

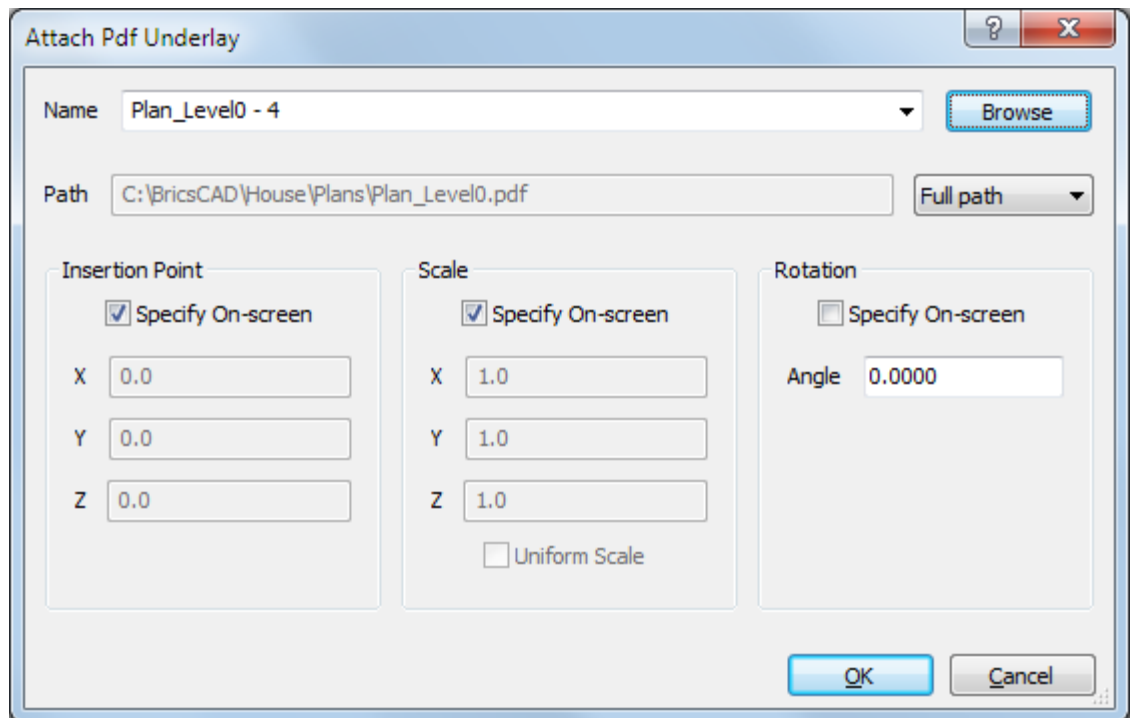
If a PDF underlay contains layers, you can control the display of these layers using the PDFLAYERS command.

The PDFCLIP command crops the display of a PDF. You can choose between a polygonal or a rectangular boundary. Each instance of the same PDF underlay file can have a different boundary.

To attach a PDF underlay

1. Do one of the following:
 - Click the *Attach pdf* tool button (📎) on the *Insert* toolbar.
 - Click the *New* tool button (📄) on the *Details* toolbar of the Drawing Explorer - PDF Underlays dialog.
 - Type *PDFattach* in the command bar, then press Enter.

The *Select PDF Underlay File* dialog opens.
2. Select the file, then double click the file or click the *Open* button on the *Select PDF Underlay File* dialog.
The *Attach PDF Underlay* dialog opens.



3. In case of a multi-page PDF, click the down arrow in the *Name* field, then select the page of the PDF you want to load.
4. Click the *Path* option button, then select either:
 - *Full path*: The full path will be searched when the drawing is loaded. If the overlay is not found there, the folder of the parent drawing is searched. If the overlay file is not found there either, "Missing or invalid reference" displays at the insertion point of the PDF overlay.
 - *Relative path*: The relative path, with respect to the folder of the parent drawing, will be searched when the drawing is loaded. If the overlay is not found there, the folder of the parent drawing is searched. If the overlay file is not found there either, "Missing or invalid reference" displays at the insertion point of the PDF overlay.
 - *No path*: Only the folder of the parent drawing is searched when the drawing is loaded. If the overlay file is not found there, "Missing or invalid reference" displays at the insertion point of the PDF overlay.
5. Choose whether you want to specify the *Insertion Point*, *Scale* and the *Rotation* angle on-screen or not.
6. Click the *OK* button.
The *Attach PDF Underlay* dialog closes.
7. Depending on the insertion options chosen in the previous step you are prompted to specify the *Insertion Point*, *Scale* and/or *Rotation* angle.

To insert a PDF underlay

1. If you want to insert a second instance of a PDF that is already attached or another page of such underlay do the following:
2. Choose *Drawing Explorer > PDF Underlays...* in the *Tools* menu.
The *Drawing Explorer - PDF Underlays* dialog opens.
3. Select the PDF underlay.
4. Do one of the following:
 - Right click and choose *Insert* in the context menu.
 - Choose *insert* in the *Edit* menu on the *Drawing Explorer - PDF Underlays* dialog.
 The *Drawing Explorer - PDF Underlays* dialog closes.

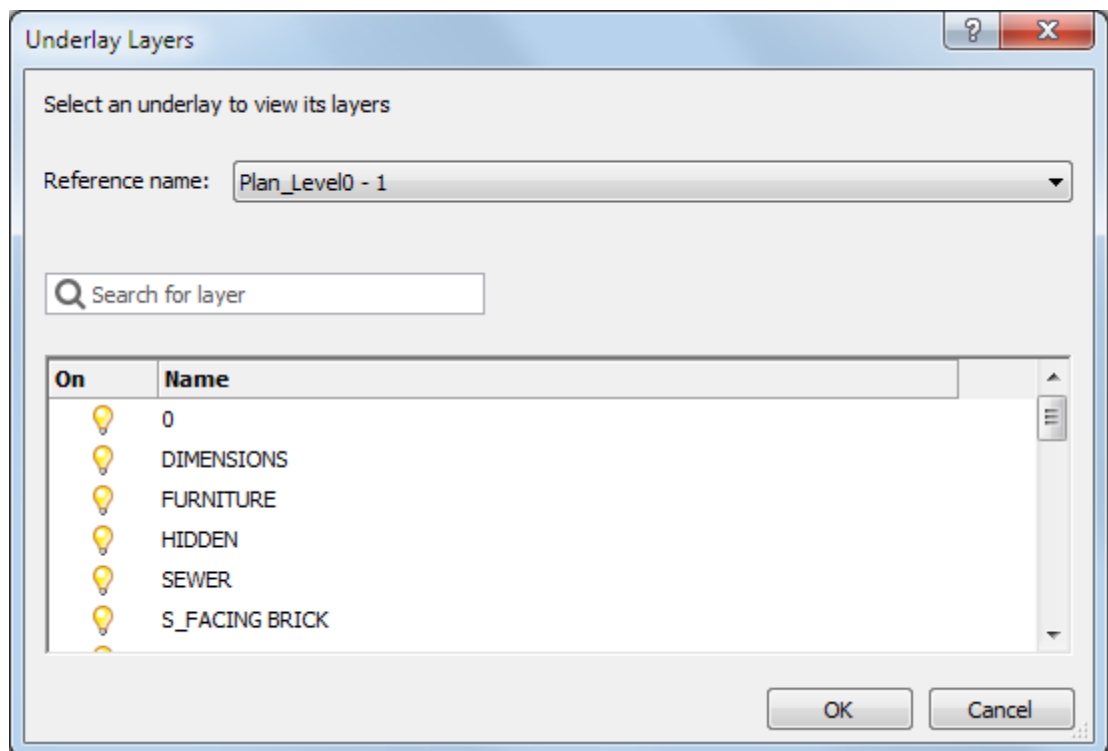
- Continue with steps 3 through 7 of the previous procedure.

To control the display of a PDF underlay

- Click the frame of the PDF underlay.
The frame of the selected PDF highlights.
The properties of the underlay display in the Properties Bar.
- Under *Misc*, click *Show underlay* in the Properties Bar, then select *Yes* or *No*.

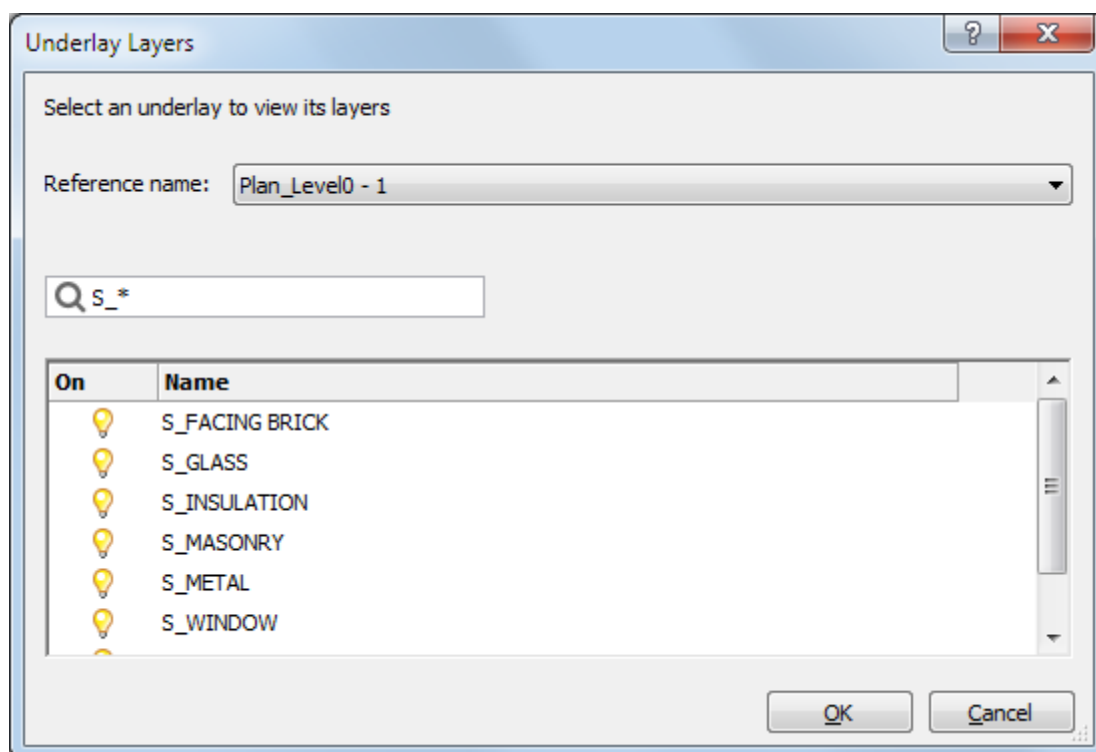
To set the layer display in a PDF underlay

- Type *PDFlayers* in the command bar, then press Enter.
The command bar reads: Select PDF underlay.
- Click the frame of the PDF underlay.
The frame of the selected PDF highlights.
The *Underlay Layers* dialog opens.

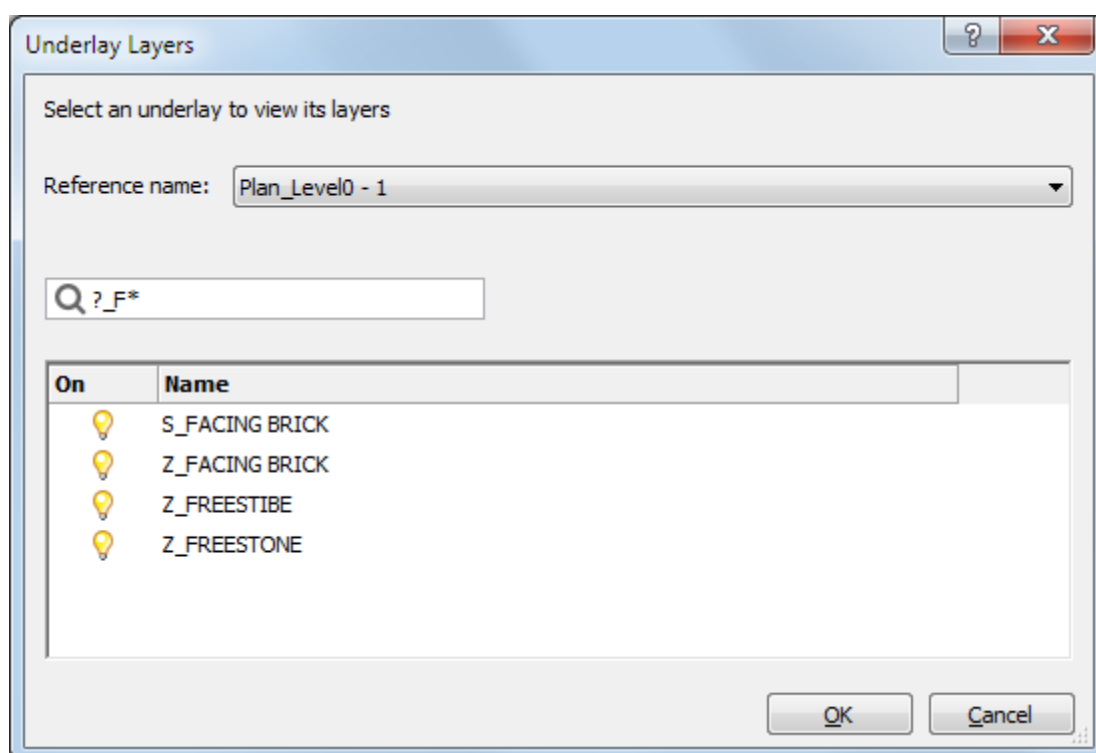


The icon in the *On* column indicates the current state of the layers On (💡) or Off (💡).

- Click the icon in the *On* column to toggle the display of a layer.
- (option) Click the *Search for layer* field, then type a layer name to search for a layer.
Use wildcard characters (? or *) to limit the number of layers in the list:



Layer names starting with 'S_' only.

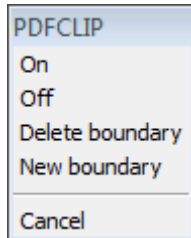


Layer names of which the second and third character is '_F' only.

5. Do one of the following:
 - Click the *Reference name* button to select another PDF underlay.
 - Click the *OK* button to stop.

To clip a PDF underlay

1. Type *pdfclip* in the command bar, then press Enter.
The command bar reads: Select PDF underlay:
2. Click the frame of the PDF underlay.
The frame of the selected PDF highlights.
The command bar reads: Enter PDF clipping option [ON/OFF/Delete/New] <New>:
A prompt menu displays:



3. Do one of the following:
 - Choose *New boundary* in the prompt menu.
 - Press Enter to accept the *New* default command option.

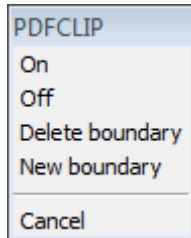
The command bar reads: Enter PDF clipping type [Polygonal/Rectangular]
<Rectangular>:
4. Do one of the following:
 - Press Enter to accept the Rectangular default command option to define a rectangular clipping boundary.
 - Type P, then press Enter to define a polygonal clipping boundary.
5. Define the clipping boundary.
The defining points must lie inside the frame of the PDF underlay. If you click outside the PDF underlay the point is placed on the PDF underlay frame.

To toggle the display of the clipped part of a PDF underlay

1. Click the frame of the PDF underlay.
The frame of the selected PDF highlights.
The properties of the underlay display in the Properties Bar.
2. Under *Misc*, click *Show clipped* in the Properties Bar, then select *Yes* or *No*.

To delete a clipping boundary

1. Type *pdfclip* in the command bar, then press Enter.
The command bar reads: Select PDF underlay:
2. Click the frame of the PDF underlay.
The frame of the selected PDF highlights.
The command bar reads: Enter PDF clipping option [ON/OFF/Delete/New] <New>:
A prompt menu displays:



3. Do one of the following:
 - Choose *Delete boundary* in the prompt menu.
 - Type D in the command bar, then press Enter.The clipping boundary is deleted and the complete PDF underlay displays.

Exploring Images

Commands: IMAGE, IMAGEADJUST, IMAGEATTACH, IMAGECLIP, IMAGEFRAME, IMAGEQUALITY

The *Image* command Inserts images in drawings through the Images Explorer.

The *Imageadjust* command adjusts the properties of images through the Properties bar.

The *Imageattach* command attaches raster images to the drawing.


The *Imageclip* command clips images.

The *Imageframe* command toggles the display of the frame around images.




The *Imagequality* command toggles the display quality of images between draft and high.

Open the Images Explorer

To open the *Images Explorer* do one of the following:

- Choose *Drawing Explorer > Images* in the *Tools* menu.
- Click the *Image Management* tool button () in the *Images* toolbar.
- Type *image* in the command bar, then press Enter.
- Select *Images* in the *Open Drawings* sub-window of the Drawing Explorer.


In the *Images Explorer* you can:

- See a preview of the attached images
- Place images () or ()
- Load / Unload inserted images
- Detach images ()

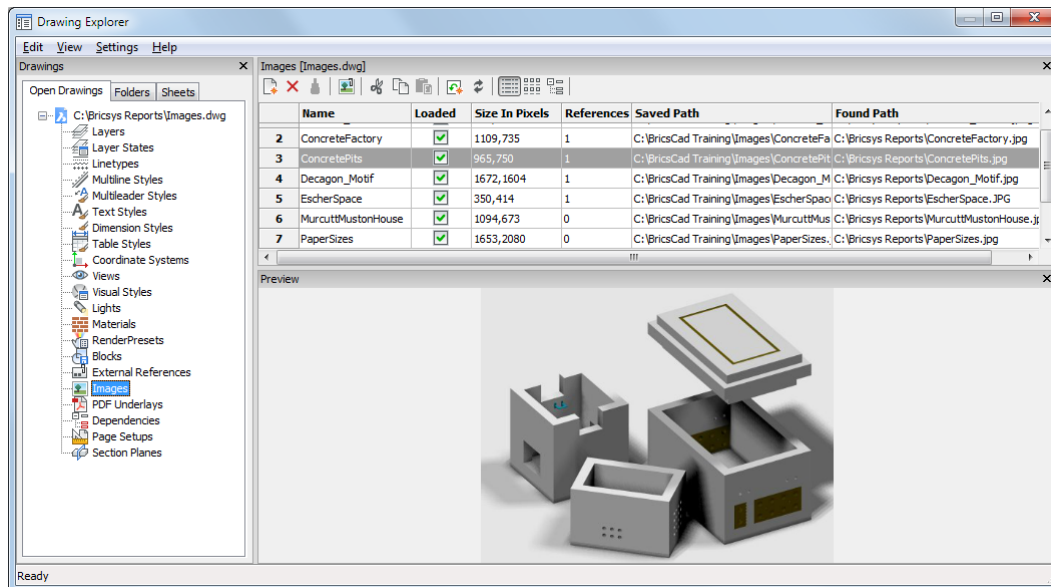
Images Explorer display options

You can choose between *Detail View* (list), *Icon View* and *Tree View* to see the images in the current drawing.


Open the Detail View of the Images Explorer

Click the *Detail View* button () in the *Drawing Explorer* toolbar. The *Detail View* button is now pressed, indicating the image details are displayed.

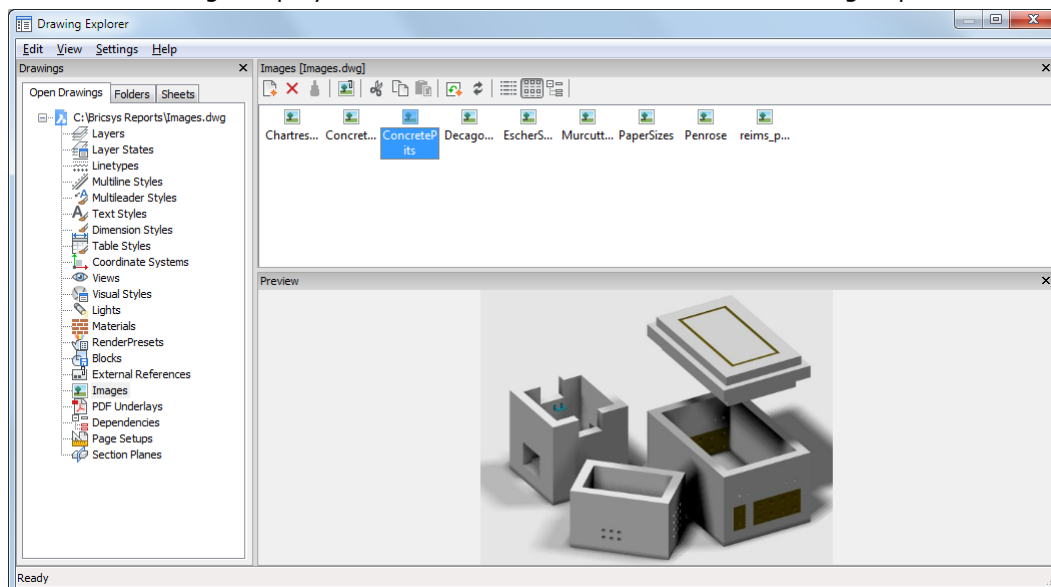
The selected image displays in the *Preview* sub-window of the *Drawing Explorer*.




Open the Icon View of the Images Explorer

Click the *Icon View* button () in the *Drawing Explorer* toolbar. The *Icon View* button is now pressed, indicating the image icons are displayed.

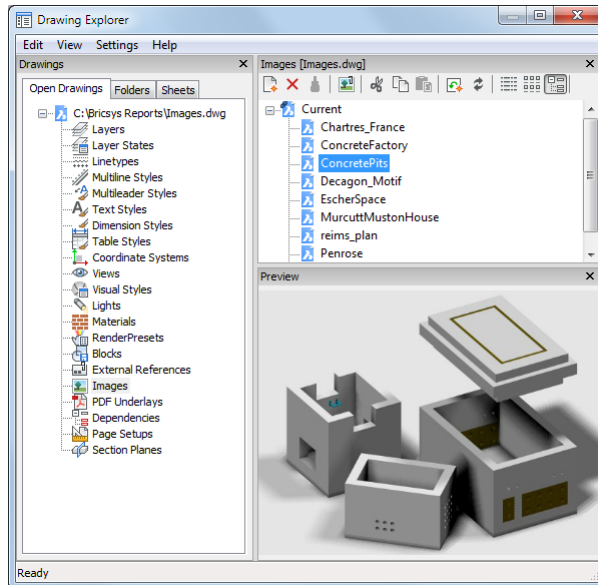
The selected image displays in the *Preview* sub-window of the *Drawing Explorer*.



Open the Tree View of the Images Explorer




Click the *Tree View* button () in the *Drawing Explorer* toolbar. The *Tree View* button is now pressed, indicating the image tree is displayed.

The selected image displays in the *Preview* sub-window of the *Drawing Explorer*.

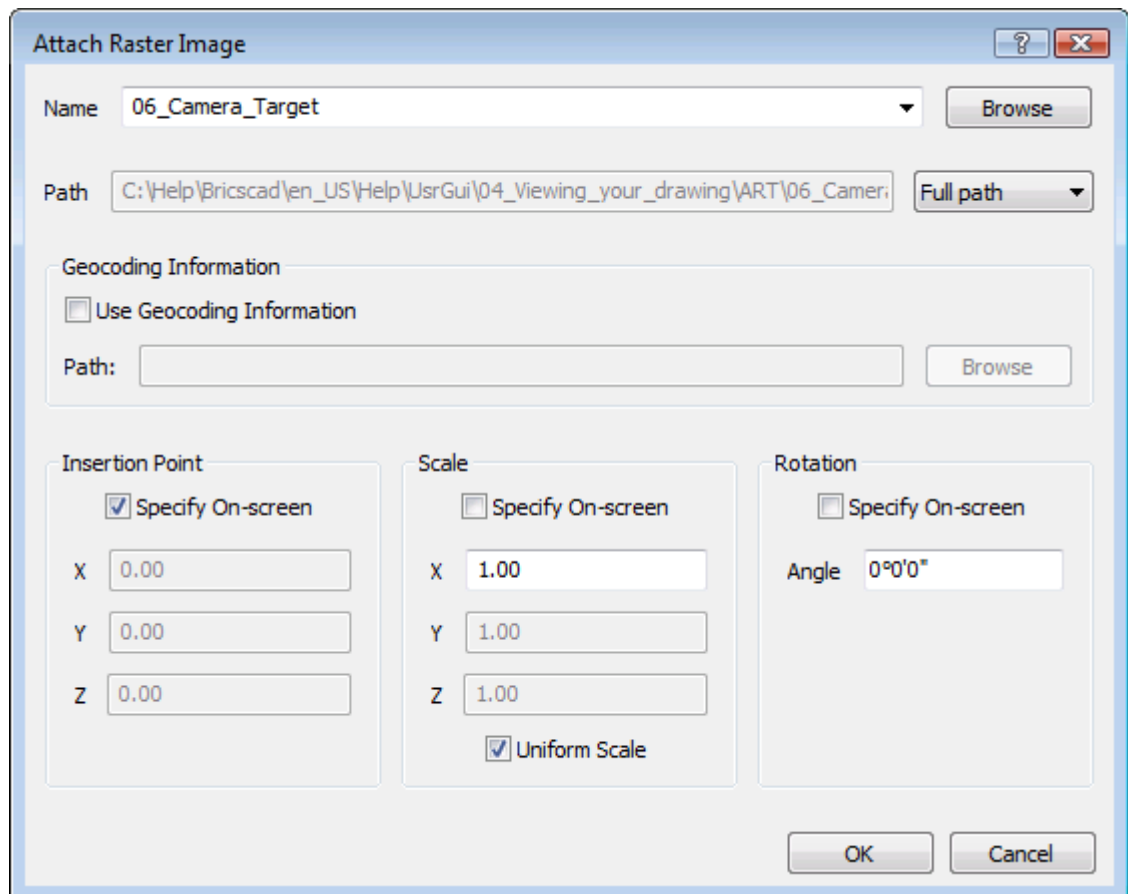


Placing images in a drawing

1. Do one of the following:

- Launch the *Image Explorer*, then click the *New* tool button () in the *Drawing Explorer - Images* toolbar.
- Launch the *Image Explorer*, select a raster image file in the list, then click the *Insert* tool button () in the *Drawing Explorer - Images* toolbar.
- Click the *Attach Raster Image* tool button () in the *Insert* toolbar.

The *Attach Raster Image* dialog opens.



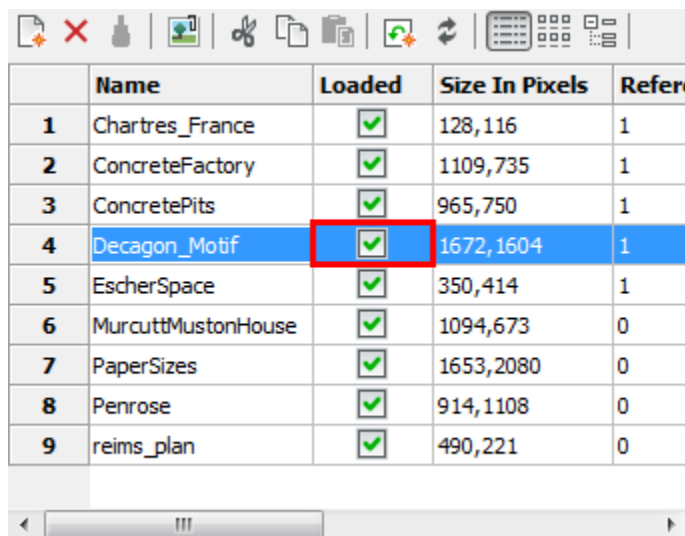
2. Do one of the following:
 - Accept the currently selected image file.
 - Click the down arrow button in the *Name* field to select a different raster image file in the list.
Raster images which are already attached are listed.
 - Click the *Browse* button to select a new raster image file in the *Select Raster Image* dialog.
3. (option) Check the ☐ , then click the *Browse* button to open an appropriate world file to be used with the selected raster image.
If the *Use Geocoding Information* option is checked, the *Insertion Point*, *Scale* and *Rotation* settings fields are not available.
4. (option) Do the following:
 - Specify an *insertion point* or select the *Specify on-screen* check box.
 - Specify *scale* or select the *Specify on-screen* check box.
 - Specify *rotation* angle or select the *Specify on-screen* check box.
5. Click the *OK* button.
6. If none of the *Specify on-screen* options is selected, the image is inserted at the desired insertion point, scale and rotation angle;
else
You are prompted to specify the insertion point, scale and/or rotation angle on-screen.

Load / Unload images



When an image is attached and then inserted in the drawing, you can unload the image to temporarily remove it. Unloaded images are still inserted in the drawing, but they no longer display. If the *Imageframe* setting is ON, the frame around the image still displays though.

Load / Unload a single image

1. Launch the *Image Explorer*.
2. For the image you want to load or unload, click the check box in the *Loaded* column.



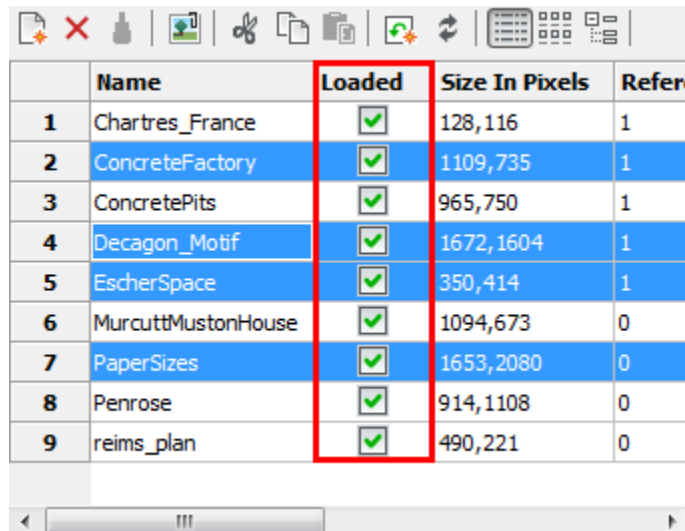
	Name	Loaded	Size In Pixels	Refer
1	Chartres_France	<input checked="" type="checkbox"/>	128,116	1
2	ConcreteFactory	<input checked="" type="checkbox"/>	1109,735	1
3	ConcretePits	<input checked="" type="checkbox"/>	965,750	1
4	Decagon_Motif	<input checked="" type="checkbox"/>	1672,1604	1
5	EscherSpace	<input checked="" type="checkbox"/>	350,414	1
6	MurcuttMustonHouse	<input checked="" type="checkbox"/>	1094,673	0
7	PaperSizes	<input checked="" type="checkbox"/>	1653,2080	0
8	Penrose	<input checked="" type="checkbox"/>	914,1108	0
9	reims_plan	<input checked="" type="checkbox"/>	490,221	0

3. If the *Regen On/Off* button () is not pressed, click the *Regen* tool () in the *Images Explorer* toolbar.

NOTE It is not necessary to select the image first in step 2.



Load / Unload multiple images

1. Launch the *Image Explorer*.
2. Select the images you want to load or unload.
3. For one of the selected images, click the check box in the *Loaded* column.



	Name	Loaded	Size In Pixels	Refer
1	Chartres_France	<input checked="" type="checkbox"/>	128,116	1
2	ConcreteFactory	<input checked="" type="checkbox"/>	1109,735	1
3	ConcretePits	<input checked="" type="checkbox"/>	965,750	1
4	Decagon_Motif	<input checked="" type="checkbox"/>	1672,1604	1
5	EscherSpace	<input checked="" type="checkbox"/>	350,414	1
6	MurcuttMustonHouse	<input checked="" type="checkbox"/>	1094,673	0
7	PaperSizes	<input checked="" type="checkbox"/>	1653,2080	0
8	Penrose	<input checked="" type="checkbox"/>	914,1108	0
9	reims_plan	<input checked="" type="checkbox"/>	490,221	0

All selected images will be loaded or unloaded simultaneously.


4. If the *Regen On/Off* button () is not pressed, click the *Regen* tool () in the *Images Explorer* toolbar.

NOTE Press and hold the Ctrl key to select multiple images.

Removing images

Since an image is first attached, then inserted in the drawing, you can delete an image, without detaching it.

Detach images


1. Launch the *Image Explorer*.
2. Select the image(s) that you want to detach.
3. Do one of the following
 - Click the *Delete* tool button () in the *Images Explorer* toolbar.
 - Right click and select *Delete* in the context menu.

NOTE Press and hold the Ctrl key to select multiple images.

Image Utilities

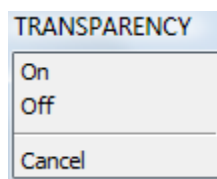
To set the image transparency

1. Do one of the following:

- Click the *Image Transparency* tool button () on the *Images* toolbar.
- Choose *Image Transparency* in the *Images* menu.
- Type *transparency* in the command bar, then press Enter.

The command bar reads: Select images.

2. Select the images you want to set the *Transparency* property of.
The command bar reads: Enter transparency mode [ON/OFF]<current mode>:
A prompt menu displays.




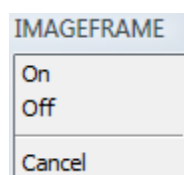
3. Do one of the following:
 - Press Enter to accept the current mode.
 - Choose *On* in the prompt menu or type *ON* in the command bar, then press Enter to switch the *Transparency* property of the selected images on.
 - Choose *Off* in the prompt menu or type *OFF* in the command bar, then press Enter to switch the *Transparency* property of the selected images off.



Transparency OFF (left) and Transparency ON (right)


Modify the Image Frame setting

1. Click the *Image Frame* tool () in the *Images* toolbar.
2. Choose *On* or *Off* in the *Imageframe* prompt menu.



The display of the image frames changes accordingly.

To clip an image

1. Make sure image frames display.
2. Do one of the following:
 - Click the *Clip Image* tool button () on the *Image* toolbar.
 - Type *imageclip* in the command bar, then press Enter.

The command bar reads: Select image:
3. Click the image frame.
The command bar reads: Enter image clipping option [ON/OFF/Delete/New] <New>:
4. Press Enter to accept the *New Boundary* default option.
The command bar reads: Enter image clipping type [Polygonal/Rectangular] <Rectangular>:
5. Press Enter to accept the *Rectangular* default option.
The command bar reads: Specify first corner
6. Click a point to define the first corner of the clipping rectangle.
The clipping rectangle displays dynamically.
The command bar reads: Specify opposite corner:
7. Click a point to define the clipping rectangle.
The image is clipped.

To toggle the clipping boundary of an image

1. Make sure image frames display.
2. Click the image frame.
The properties of the image display in the Properties Bar.
3. Under *Misc / Show clipped* choose *Yes* or *No*.

Printing

Commands: PLOT, PRINT, QPRINT, PAGESETUP, PUBLISH

The *Plot* command plots drawings to printers and to files, through prompts at the command bar; this command is meant for scripts and routines.

The *Print* command prints drawings to plotters and files through a dialog box.

The *Qprint* command prints the drawing with the default plot configuration, without displaying the Print dialog box (short for "quick print").

The *PageSetup* command creates and edits page setups for plotting drawings in the Drawing Explorer.



Printing (Internet connection needed)

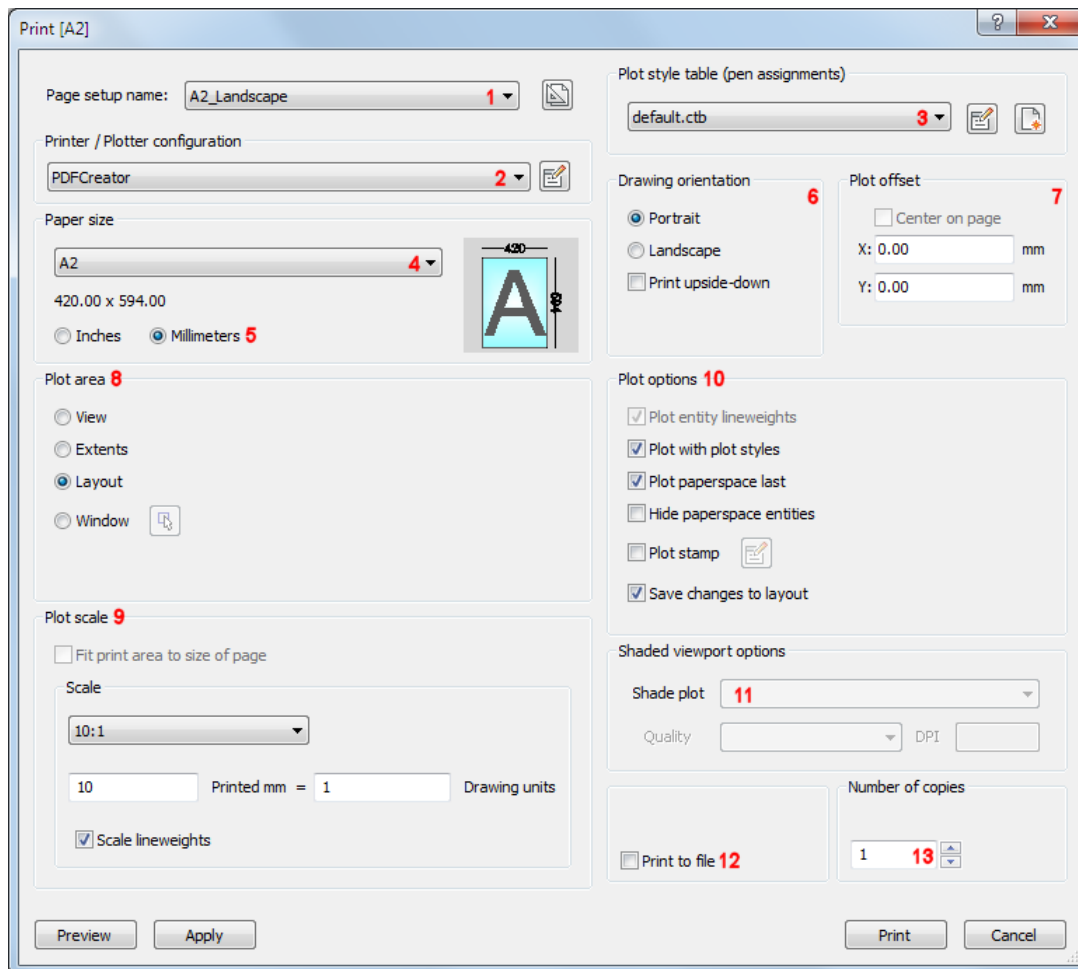
A standard print procedure consists of the following steps:

- Choose either a Layout tab or the Model tab.
- Do one of the following to open the *Print* dialog:
 - Choose *Print* in the *File* menu.
 - Type *print* in the command bar, then press Enter.
 - Press *Ctrl + P*
- Select a Page Setup name. (1)

The following steps are optional and only necessary if you want to use overrides on the selected page setup definition.

- Choose a Printer or Plotter Configuration. (2)
- Choose a Plot style table. (3)
- Choose a Paper Size. (4)
The image reflects the position and orientation of the drawing on the selected paper size.
- Choose the Paper Units. (5)
- Choose the Drawing Orientation. (6)
- Define the Plot offset. (7)
- Specify the Area to plot. (8)
- Define the Plot Scale. (9)
- Check the Plot Options. (10)
- When printing model space, choose a Shade Plot mode. (11)
This option is disabled when printing a paper space layout.
- Check/uncheck the *Print to file* option (12)
- Specify the Number of copies. (13)
- Click the *Preview* button to see a preview of the plot output.
- When in a layout, click the *Apply* button to update the layout.
- Click the *Print* button to start printing.

NOTE All print settings and options are saved in the *Model* and each of the *Layouts*.



To open the Print dialog

Do one of the following:

- Choose *Print...* in the File menu.
- Type *print* in the command bar, then press Enter.

NOTE If you click the *Print* (🖨️) button in the *Standard* toolbar, the *Qprint* command is launched, which will start printing the currently selected view tab using previously saved print settings.

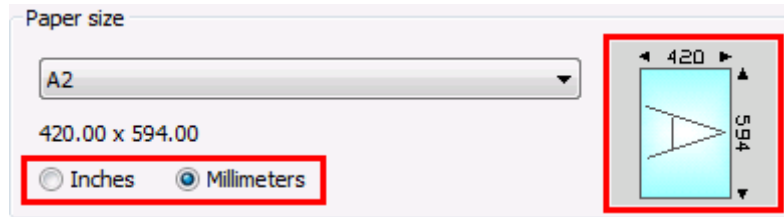
To choose the paper size

Press the down arrow, then select the paper size in the *Paper Size* list.

The available sizes depend on the selected printer.

To choose the paper units

Select either *Inches* or *Millimeters*.



The paper size image indicates the size, position and orientation of the plot area on the selected paper size.

NOTE The *Scale* and *Plot offset* setting units are changed according to the chosen *Paper units*.

To choose the paper orientation

Select either *Portrait* or *Landscape*.

- *Portrait*: The drawing or layout x-axis is aligned with the shortest edge of the selected paper size.
- *Landscape*: The drawing or layout x-axis is aligned with the longest edge of the selected paper size.

The paper size image changes accordingly.

When in a layout, click the *Apply* button to update the display.

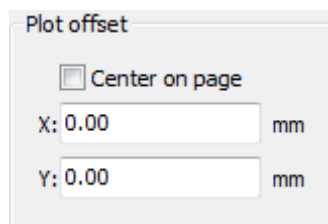
To define the plot offset

Select either *Center on page*

or

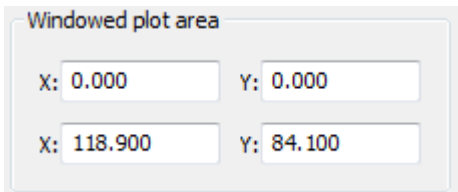
type the plot offset in the X: and Y: fields.

The offset is calculated from the bottom left corner of the paper.



When in a layout, click the *Apply* button to update the display.

To specify the area to plot

Option	Result
View	Prints the portion of the drawing that displays in the current viewport or a saved view.
Extents	Prints all entities in the drawing
Limits	Prints to the limits defined in the drawing This option is only available when printing in Model space.
Layout	Prints the current layout. This option is only available when printing in Paper space.
Window	<p>Prints the portion of the drawing contained in a user defined window, maintaining the aspect ratio of the windowed area to the drawing.</p> <p>Click the <i>Select print area</i> button (), then specify the two corners of the print area in the drawing.</p> <p>or</p> <p>Type the the X- and Y-coordinates of the corners of the print area in the X: and Y: fields.</p> 

To define the plot scale

NOTE The SCALELISTEDIT command allows to edit the list of available scales in the *Print* dialog and the *Standard Scale* property of a paper space viewport.

... when printing in paper space:

If you print in paper space, the final scale of the plot output is the product of the viewport scale and the plot scale.

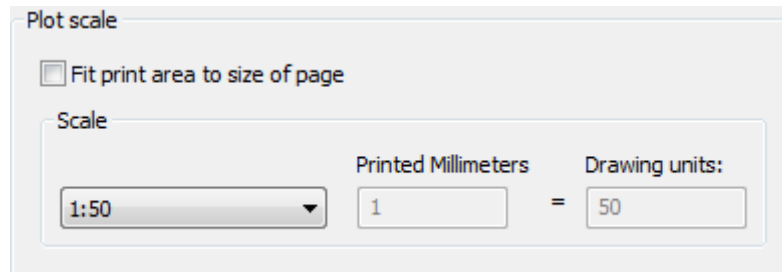
In order to keep control over the scale of the plot output:

- Specify the width and height of the viewports in drawing units.
- Set the scale of each viewport to the scale you want in the plot output.
- The *plot scale* reflects the relationship between paper units and drawing units. (see table below)

Paper units	Drawing units	Plot scale	Printed Millimeters or Inches	Drawing Units
mm	mm	1:1	1	1
mm	cm	10:1	10	1
mm	m	1000:1	1000	1
inches	inches	1:1	1	1
inches	feet	12:1	12	1

... when printing in model space:

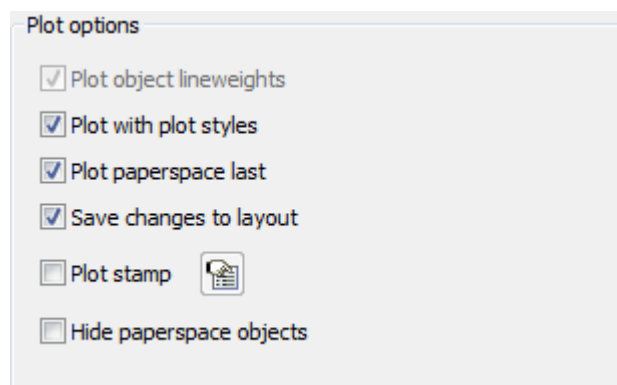
If you print in model space, the scale of the plot output equals the plot scale, on condition your drawing units equal the paper units. Otherwise you need to multiply the plot scale with respect to the relationship between paper units and drawing units. E.g. if you want to plot at a 1/50 scale in a drawing of which 1 DU = 1 cm, the *plot scale* = 1/5, if 1 DU = 1 m the *plot scale* = 20/1 or 1/0.05



The 'Plot scale' dialog box contains a checkbox for 'Fit print area to size of page'. Below it is a 'Scale' section with a dropdown menu showing '1:50'. To the right of the dropdown are two input fields: 'Printed Millimeters' with the value '1' and 'Drawing units' with the value '50', separated by an equals sign.

To set the plot options

Click to toggle the *plot options* On / Off.

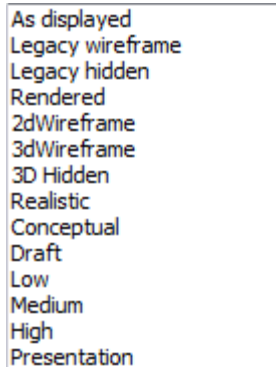


The 'Plot options' dialog box lists several options with checkboxes: 'Plot object lineweights' (checked), 'Plot with plot styles' (checked), 'Plot paperspace last' (checked), 'Save changes to layout' (checked), 'Plot stamp' (unchecked, with an icon), and 'Hide paperspace objects' (unchecked).

Option	Result
Plot object lineweights	If <i>Plot with plot styles</i> is off, object lineweights are plotted.
Plot with plot styles	Settings of the assigned STB or CTB file are used.
Plot paperspace last	Entities in Model space are plotted first, paperspace entities last.
Save changes to layout	All changes made in the <i>Print</i> dialog are saved in the layout. This option toggles the <i>Save changes to layout</i> variable
Plot stamp	Adds a header and / or footer. Click the <i>Edit plot stamp</i> button (📄✎) to define the plot stamp.
Hide paperspace objects	Removes hidden lines from 3D entities in paper space. This option is disabled when printing model space.

To set the shade plot mode

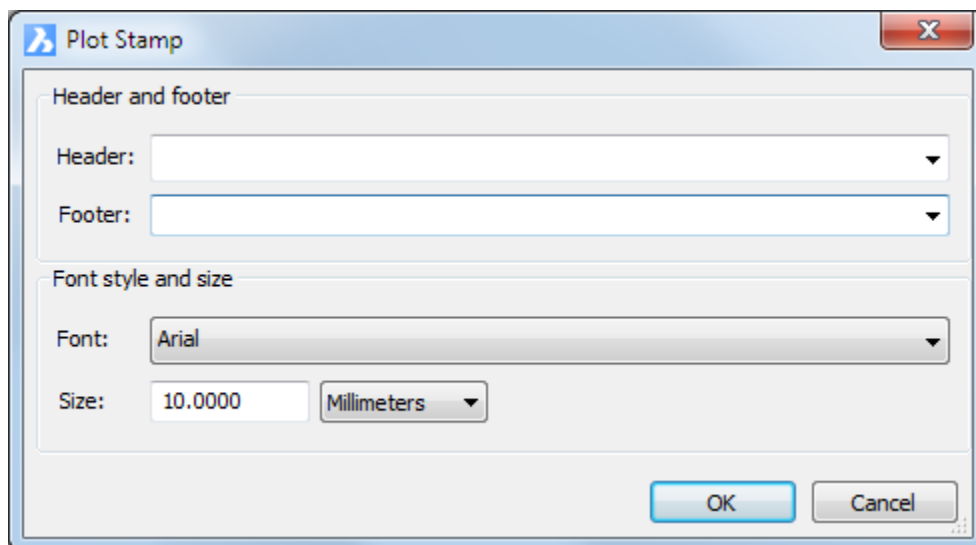
Under *Shade viewport options*, click the list button, then select an option:



NOTE This option applies to model space printing only.
The *Shade Plot* mode of a paper space viewport is defined in the *Shade Plot* property of the viewport.

To define the plot stamp

1. Click the *Edit plot stamp* button (📄✎).
The *Plot stamp* dialog opens.



2. In the *Header* field, do one of the following:
 - Type the header text.
 - Click the down arrow button, then choose a variable in the drop down list.
 - Type one or more variable names.
 - Leave the field empty.
3. In the *Footer* field, do one of the following:
 - Type the footer text.
 - Click the down arrow button, then choose a variable in the drop down list.
 - Type one or more variable names.
 - Leave the field empty.

4. Select a font in the *Font* list button.
5. Enter a value in the *Font Size* field
6. Set the unit type to *Millimeters* or *Inches*.
7. Click the *OK* button to confirm.

The plot stamp settings are saved to a series of user preferences in the current user profile, which can be edited in the Settings dialog under *Program Options / Plot and Publish / Plot Stamp*:

Plot stamp	
Header	
Footer	
Font Style	Arial
Font Size	10
Units	[1] Millimeters

Variables

Name	Description
\$Name	Adds the name of the current user, as defined in the <i>User Name</i> field of the License Information.
\$Company	Adds company name of the current user, as defined in the <i>Company</i> field of the License Information.
\$Date	Adds the current date.
\$DateTime	Adds the current date and time.
\$DwgName	Adds the file name and path of the drawing.

NOTE The header and footer text consists of three parts (left, center and right), separated by commas.

By default the header and footer text is centered.

The header and footer text can be outlined left center and/or right by separating the text with commas.

e.g. *\$Name,\$Company,\$Date* places your name left, company name centered and the current date at the right.

Type a space before one of the commas to leave one of the parts empty, thus outlining the header or footer left or right.

\$Company, , (\$Company,<space>,<space>) outlines the company name left.

,,\$Company (<space>,<space>,\$Company) outlines the company name right.

To specify the number of copies

Do one of the following:

- Type the desired number in the *Number of copies* field.
- Click the up-arrow and the down-arrow buttons to specify the number of copies.

Number of copies:	1	▲ ▼
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To see a preview of the plot output

1. Do one of the following:
 - Click the *Preview* button on the *Print* or *Page Setup* dialogs.
 - Choose *Print Preview...* in the *File* menu.
 - Type *preview* in the command bar, then press Enter.
2. (option) Use the mouse scroll wheel to zoom. Double click the middle mouse button to zoom extents.
3. To close the *Print Preview* window, do one of the following:
 - Click the *Close* button to return to the drawing window.
 - Click the *Print Settings...* button to open the Print dialog.
 - Click the *Print* button to start printing.

Plotter Configuration

Command: PLOTTERMANAGER

Starting from BricsCAD V5, configurations for Windows system printers are stored in plotter configuration files (.PC3 files). Unlike in AutoCAD®, PC3 plotter configuration files in BricsCAD™ apply to Windows system printers only.

In a plotter configuration file, you override one or more settings of the system printers installed on your computer. You can configure BricsCAD for many plotting/printing devices and keep multiple configurations for each single device.

Each plotter configuration file contains information such as:

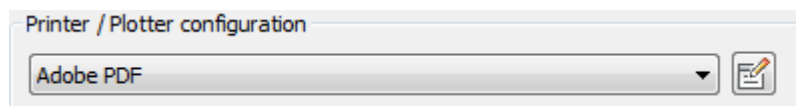
The device driver and model

The output port to which the device is connected

Various device-specific settings (not supported on the Linux platform).

PC3 files are saved in the *Plotconfig* subfolder of your Roamable root folder.

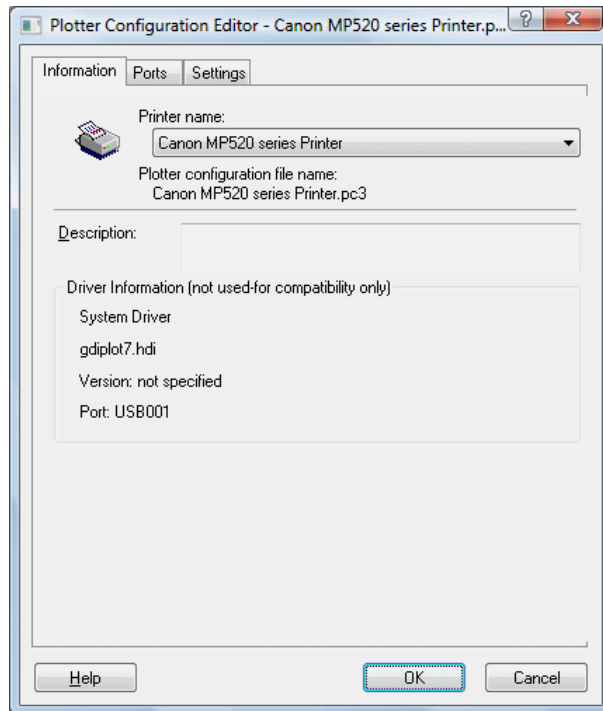
After creating a PC3 file, it is available in the list of plotter configuration names in the *Printer / Plotter configuration* list on the *Print* or *Page Setup* dialog boxes.



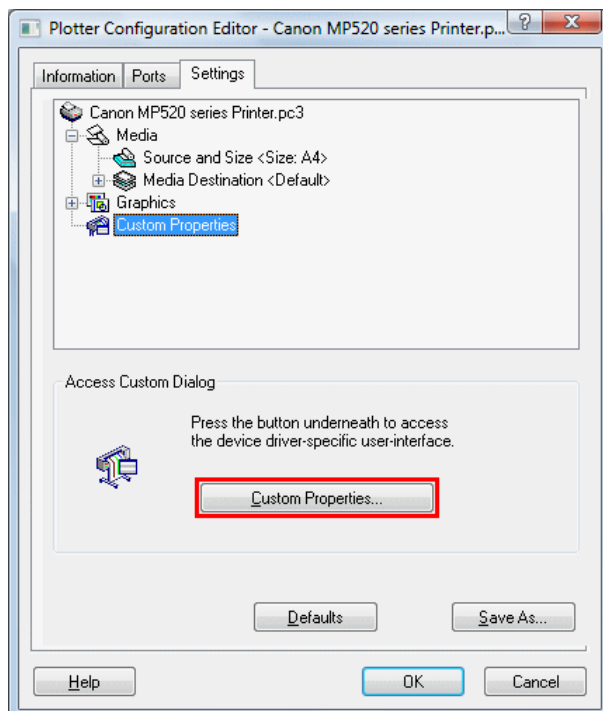
To create a plotter configuration file

1. Choose *Plotter Manager...* in the *File* menu, then double click *Create a Plotter Configuration* shortcut.

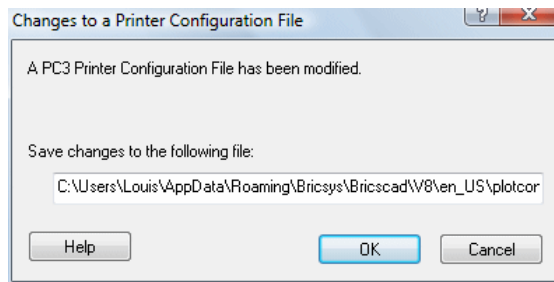
The *Plotter Configuration Editor* dialog opens.



2. In the *Printer name* list, select the printer you want to create a plotter configuration (PC3) for.
3. Click the *Settings* tab on the *Plotter Configuration Editor* dialog.



4. Click the *Custom Properties...* button.
The *<Selected Printer> Properties* dialog opens.
The layout of this dialog is different for each printer.
Custom Properties are not supported on the Linux platform.
5. On the *Properties* dialog of the selected printer, adjust the printer settings, then click the *OK* button to confirm.
The *<Selected Printer> Properties* dialog closes.
6. Click the *OK* button on the *Plotter Configuration Editor* dialog.
The *Changes to a Printer Configuration File* dialog opens.



7. (option) Adjust the PC3 file name.
The default name is *<Selected Printer>.pc3*.
8. Click the *OK* button on the *Changes to a Printer Configuration File* dialog.
The PC3 file is created.

NOTE If you want to create multiple plotter configurations for the same printer, you must adjust the default name in step 7. If you accept the default name, the existing plotter configuration will be overwritten.

To edit a plotter configuration

1. Do one of the following:
 - Choose *Page Setup* in the *File* menu.
 - Type *pagesetup* in the command bar, then press Enter.
2. Select a plotter configuration (PC3) in the *Printer / Plotter configuration* list.
3. Click the *Edit Plotter Configuration* button (📄✎) on the *Print* or *Page Setup* dialogs.
The *Plotter configuration editor* dialog opens.
4. Proceed in the Create a plotter configuration procedure starting from step 3.

NOTE If you select a system printer in step 2, a new plotter configuration is created for the selected printer.

When you click the *Edit Plotter Configuration* button (📄✎) in step 3, the *Properties* dialog of the selected printer opens first.

After clicking the *OK* button on the *Properties* dialog the *Plotter configuration editor* dialog opens.

You can then complete the Create a plotter configuration procedure.

To assign a plotter configuration

1. Choose *Model* or the *Layout* to which you want to assign a specific plotter configuration file.
2. Do one of the following:
 - Choose *Page Setup* in the *File* menu.
 - Type *pagesetup* in the command bar, then press Enter.
3. Choose one of the following from the *Printer / Plotter configuration* list:
 - A system printer
 - A configuration file (PC3 file)
 - None (uses Default)

4. Click the *OK* button.

NOTE When BricsCAD™ cannot find the plotter configuration file assigned to the drawing, it changes the configuration file assigned to your drawing to 'None (uses Default)'. When you print the drawing, the settings of the default printer will be used instead.

When choosing the 'None (use Default)' option, a plotter configuration file *Default.pc3* will be used. It contains the plot settings found in the registry at the moment it was first created.

Plot Style tables

Plot Style Tables are used in BricsCAD to control printing attributes.

A plot style table is a collection of plot styles assigned to a Layout or the Model tab. There are two types of plot style tables:

Color-dependent plot style tables (CTB = Color Table)

Named plot style tables. (STB = Style Table)

Although each layout may use a different plot style table, each drawing may have only one type of plot style table type assigned to it. All layouts in a drawing must use all CTB files or all STB files exclusively. One drawing may not mix and match these file types. The type of plot style used in a drawing is assigned when the drawing is created.

If you create a new drawing from a template, the template's plot style table type will apply to the new drawing.

If you create a new drawing using a wizard, you will be asked which type of plot style table you want to assign.

If you create a drawing from scratch, it depends on the status of the *BASEFILE* and *PSTYLEPOLICY* variables whether *CTB* (Color table) or *STB* (Named plot style table) will be used to set up plotter configurations in the newly created drawing.

Color-dependent plot style tables (CTB file)

The color-dependent plot styles tables use an entity's color to determine characteristics such as lineweight. Use color-dependent plot styles to ensure that all entities that share the same color are plotted the same way. While you can edit plot styles in a color-dependent plot style table, you cannot add or delete plot styles. There are 255 plot styles in a color-dependent plot style table, one for each BricsCAD color.

Named plot style tables (STB file)

The named plot styles tables contain user-defined plot styles. When you use a named plot style table, entities that have the same color may be plotted differently, based on the plot style assigned to the entity. A named plot style table can contain as many or as few plot styles as required. Named plot styles can be assigned to entities or layers, in the same way that linetype and color are assigned to entities. An entity whose plot style is set to *BYLAYER* inherits the plot style assigned to its layer.

Predefined plot styles

When installing BricsCAD, the following plot style tables will be installed in the *Plotstyles* subfolder of your Roamable root folder.

BricsCAD.ctb (255 colors)

BricsCAD.stb (sample style table)

Default.ctb (255 colors)

Default.stb (sample style table)

Monochrome.ctb (all colors plot as black)

Monochrome.stb (all colors plot as black)

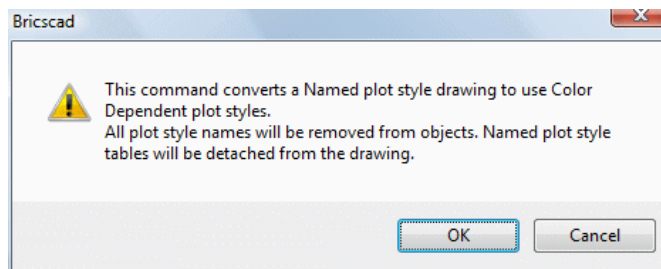
Changing Plot Style Table Types

Commands: CONVERTPSTYLES and CONVERTCTB

A common problem is that Named Plot Style tables (STB files) are inadvertently assigned to a drawing and a user wants to assign Color-Dependent Plot Style tables (CTB files) instead. Or vice versa for a user who wants to use Named Plot Styles.

To convert a drawing from STB to CTB file type

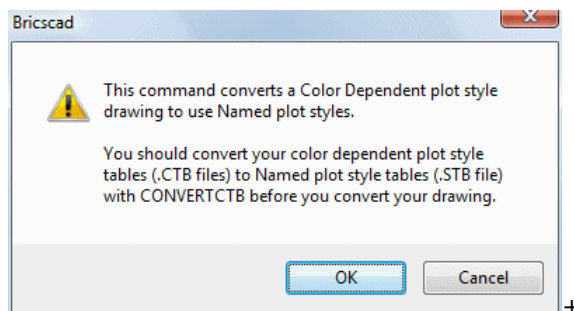
1. Type *convertpstyles* at the command prompt, then press Enter.
An alert box displays:



2. Click the *OK* button.
The drawing is converted.

To convert a drawing from CTB to STB file type

1. Type *convertpstyles* at the command prompt, then press Enter.
An alert box displays:

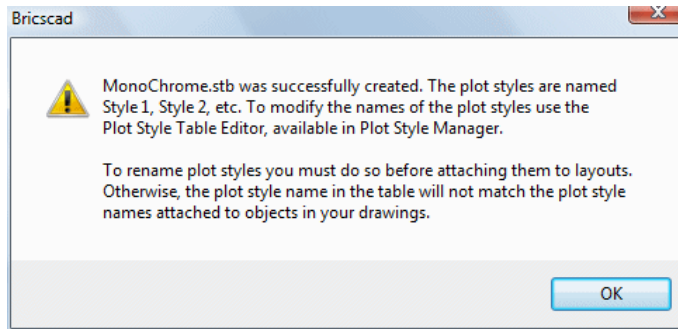


2. Do one of the following:
 - Click the *Cancel* button and convert your CTB files first.
The drawing is not converted.
 - Click the *OK* button.
The *Select style table file* dialog displays.
3. Select a *.stb file created by the *CONVERTCTB* command, then click the *OK* button.
The drawing is converted.

To convert a Color-dependent plot style table (CTB) to a Named plot style table (STB)

1. Type *convertctb* at the command prompt, then press Enter.
An *Open File* dialog opens, listing the CTB-files in the *Plotstyles* subfolder of your Roaming root folder.

2. Select the CTB-file you want to convert.
3. Click the *Open* button.
The Open File dialog now lists the STB-files in the *Plotstyles* subfolder of your Roaming root folder.
4. Do one of the following:
 - Click the *Save* button to accept the default file name.
 - Type a name in the File name field, then click the *Save* button.



The Plot Style Table editor

The *Plot Style Table Editor* can be opened, even if BricsCAD is not open.

To open the Plot Style Table editor

Do one of the following:

- Click the *Edit Plot Style* button (📄✎) on the *Print* dialog.
- Choose *Plot Style Manager...* in the File menu, then double click a *CTB* or *STB* file.
- In a *Windows Explorer* window, double click a *CTB* or *STB* file.

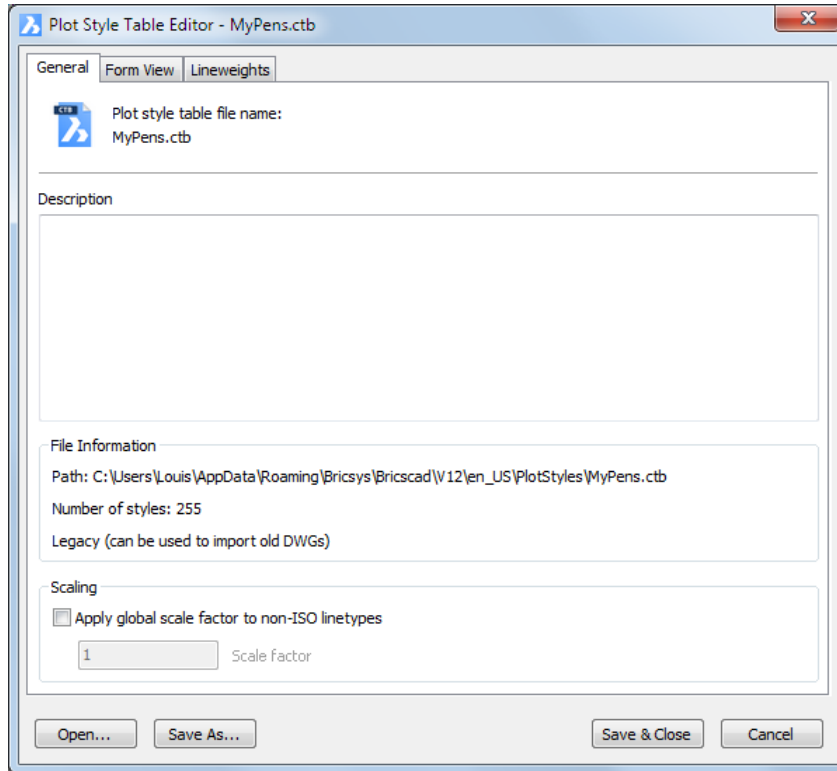
The *Plot Style Table Editor* displays the plot styles contained in the specified plot style table.

The Plot Style Table Editor includes the following tabs:

- The *General* tab
- The *Form View* tab
- the *Lineweights* tab

The General tab

The *General* tab lists general information about the plot style table.

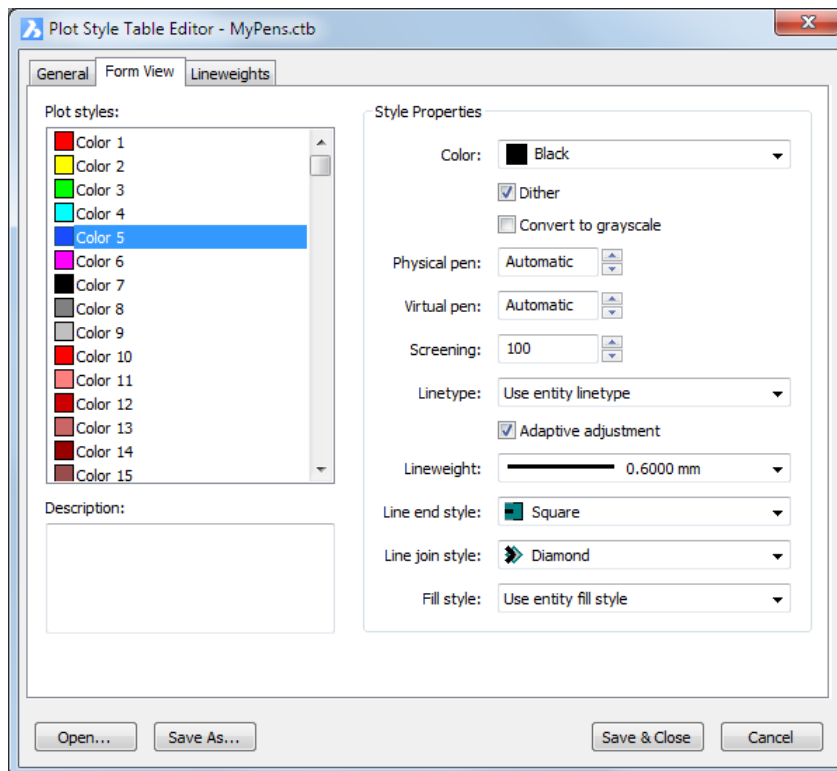


Name	Description
Plot style table file name	Displays the name of the plot style table file you are editing.
Description	Displays a description you want to include about the plot style table.
File Info	Displays information about path, number of plot styles and the version number of the Plot Style Table Editor. Note: A color-dependent plot style table always contains 255 plot styles.
Apply global scale factor to non-ISO linetypes	Specifies whether or not you want to scale all non-ISO linetypes in the plot styles of entities controlled by the plot style table.
Scale factor	Specifies the factor by which to scale non-ISO linetypes and fill patterns.

The Form View tab

The *Form View* tab lists the plot styles contained in the plot style table and their settings. Plot styles are style overrides for your drawings that occur during plotting.

The first plot style in a *Named Plot Style table* is *NORMAL* and represents an entity's default properties (no plot style applied). You cannot modify or delete the *NORMAL* style.



Property	Description
Plot styles	<p>Displays the names of plot styles contained in the selected plot style table.</p> <p>Color-dependent plot style table:</p> <p>Always contains 255 styles (1 for each color). They are tied to entity color. You cannot add nor delete a plot style. You cannot rename a plot style.</p> <p>Named plot style table:</p> <p>Contains one or more plot styles. You can add or remove plot styles, except for the <i>Normal</i> style, which cannot be deleted. Plot styles can be renamed, except for the <i>Normal</i> style, which cannot be renamed.</p>
Description	Displays a description for the selected plot style.
Color	<p>Specifies the plotted color for an entity. The default setting for plot style color is <i>Use Object Color</i>. If you assign a plot style color, the color overrides the entity's color in the plot output.</p> <p>Choose <i>Other...</i> to select one of 255 colors from the <i>Select Color</i> dialog box.</p> <p>Choose <i>True color</i> to select a color from the <i>Color</i> dialog box. If the plot device does not support the color you have specified, the nearest available color will be used. When you have a monochrome device, <i>black</i> will be used.</p>
Dither	Depending on the capabilities of your plotter, dithering approximates the colors with dot patterns. When this option is not active, the colors are

	<p>mapped to the nearest color, resulting in a smaller range of colors when plotting.</p> <p>Dithering is available only whether you select the object's color or assign a plot style color.</p> <p>By default, this option is active. Setting it to Off has no effect in BricsCAD.</p>
Grayscale	Depending on the capabilities of your plotter, converts the colors to grayscale in the plot output.
Pen #	Specifies a pen to use when plotting entities that use this plot style. You can select a pen number from 1 to 32.
Virtual pen #	<p>Specifies a virtual pen number between 1 and 255.</p> <p>Choose <i>Automatic</i> to specify that BricsCAD should make the virtual pen assignment from the BricsCAD Color Index.</p> <p>The virtual pen setting in a plot style is used only by non-pen plotters and only if they are configured for virtual pens (select <i>255 Virtual Pens</i> under <i>Color Depth</i> under <i>Vector Graphics</i> on the <i>Settings</i> tab in the <i>Plotter Configuration Editor</i>).</p>
Screening	<p>Specifies the color intensity of the plot on the paper.</p> <p>The valid range is 0 through 100.</p> <p>If you select 100 the drawing will be plotted with its full color intensity. In order for screening to work, the <i>Dithering</i> option must be active.</p>
Linetype	<p>Displays a list with an example and a description of each linetype.</p> <p>The default setting for plot style linetype is <i>Use Object Linetype</i>.</p> <p>The selected linetype overrides the entity's linetype in the plot output.</p>
Adaptive	<p>Adjusts the scale of the linetype to complete the linetype pattern.</p> <p>You can activate this property if a complete linetype pattern is more important than a correct linetype scaling.</p>
Lineweight	<p>Displays a sample of the lineweight as well as its numeric value.</p> <p>You can modify an existing lineweight if the one you need is not available.</p> <p>The default setting for plot style lineweight is <i>Use Object Lineweight</i>.</p> <p>The selected lineweight overrides the entity's lineweight in the plot output.</p>
Line end style	<p>Displays a list of several end styles for the lines.</p> <p>By default, the option <i>Use Object End Style</i> is selected.</p> <p>If you select a line end style, the object's line end style will be overridden in the plot output.</p>
Line join style	<p>Displays a list of several join styles for the lines.</p> <p>By default, the option <i>Use Object Join Style</i> is selected.</p> <p>If you select a line join style, the object's line join style will be overridden in the plot output.</p>
Fill style	<p>Displays a list of several fill styles.</p> <p>By default, the option <i>Use Object Fill Style</i> is selected.</p> <p>If you select a fill style, the object's fill style will be overridden in the plot output.</p>

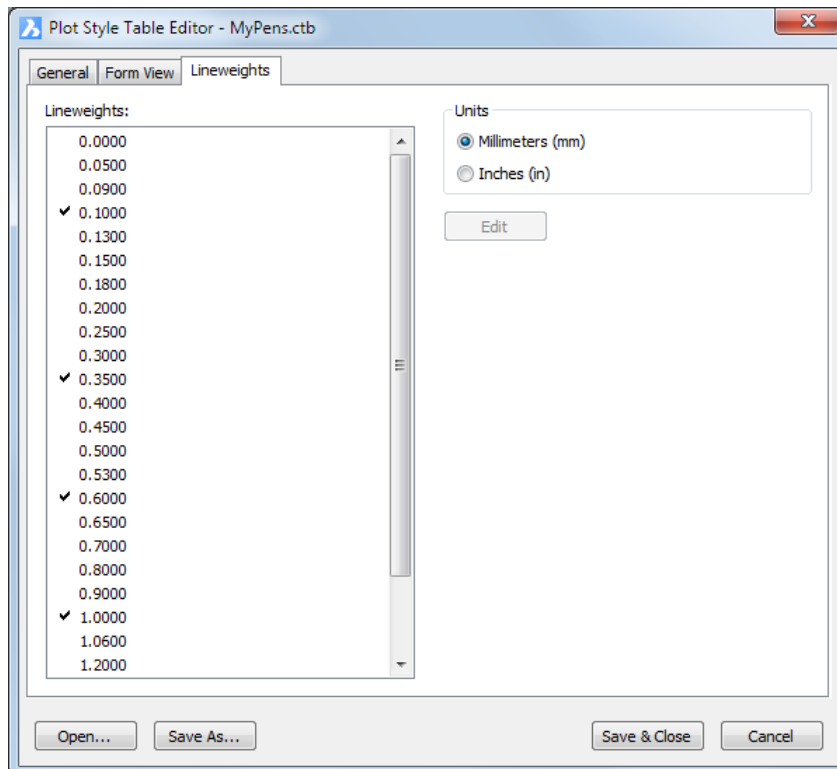
NOTE Objects which are created in *true color* are always plotted using their own object properties (color, lineweight, ...).

The Lineweights tab

The *Lineweights* tab lists the available lineweights.

Lineweights with a tick are used in the *Lineweight* field on the *Form View* tab.

To edit a lineweight, select the lineweight in the list, then click the *Edit* button.



Assigning plot style tables

In BricsCAD you can assign plot style tables to model space and to each of the layouts in a drawing. By assigning different plot style tables to each layout in your drawing, you can control how entities in the layout are plotted. The plot style table affects both model space and paper space entities. To plot the drawing without applying plot style properties, choose 'None (use Default)' from the list of plot style tables.

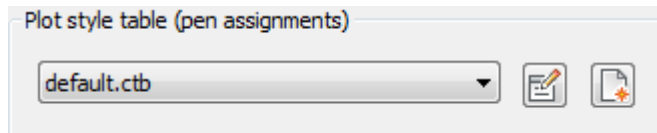
In STB-type drawings, each entity in the drawing either is assigned a plot style explicitly or *By Layer*.

Missing plot style table

If a plot style table is not found, the text "(missing)" is added to the plot style table name. When you print the drawing, the settings of the default plot style file Default.ctb or Default.stb will be used instead.

To assign a plot style table

1. Choose *Model* or the *layout* to which you want to assign a different plot style table.
2. Do one of the following:
 - Choose *Print...* in the *File* menu.
 - Type *print* in the command bar, then press Enter.
3. Select a *Plot style table* from the list box in the *Plot Style Table (pen assignments)* section.

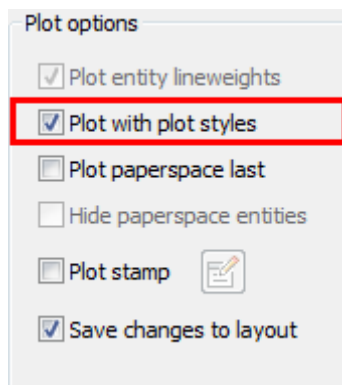


4. Click the *OK* button.

Toggling the plot style table on/off

When you turn off plot style tables, entities print according to their own properties. However, all of the plot style information is saved so you can easily turn on plot styles again. Actual plot style table files are not deleted, and for drawings that use named plot style tables, entities and layers retain their assigned plot styles.

1. Choose *Model* or the *layout* in which you want to turn the plot style table on/off.
2. Do one of the following:
 - Choose *Print...* in the *File* menu.
 - Type *print* in the command bar, then press Enter.
3. Click the *Plot with plot styles* plot option.



The Plot Styles Manager

Commands: STYLESMANAGER

The *Plot Styles Manager* is a window that lists plot style table files (CTB and STB files).

The Plot Styles Manager allows you to:

Create Color-dependent Plot Style (CTB) and Named Plot Style files (STB) using the *Create a Plot Style Table* wizard.

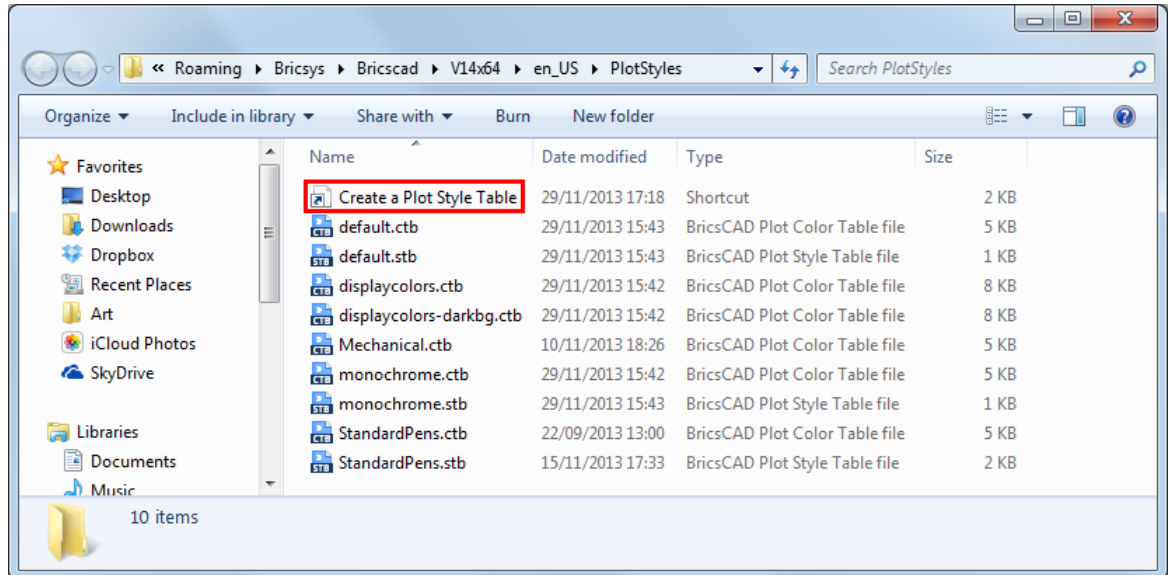
Edit existing Plot Style Table files.

To open the Plot Styles Manager

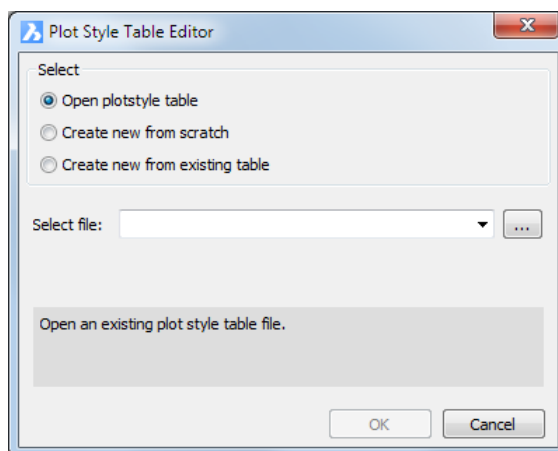
- Do one of the following:
- Choose *Plotstyle manager...* in the *File* menu.
 - Type *stylesmanager* in the command bar then press Enter.

To create a new plot style table

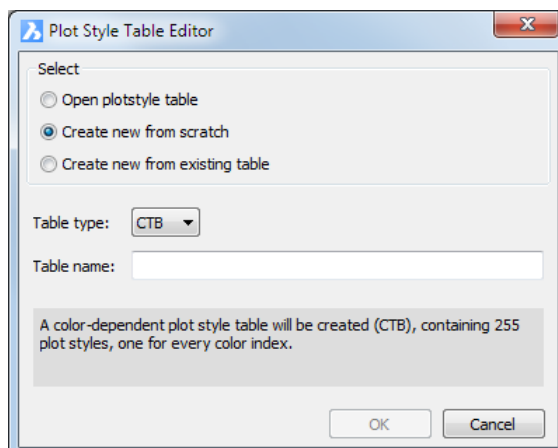
1. Open the Plot Styles Manager.
2. Double click the *Create a Plot Style Table* shortcut.



The *Add Plot Style Table Editor* dialog opens.



3. Choose *Create new from scratch*.

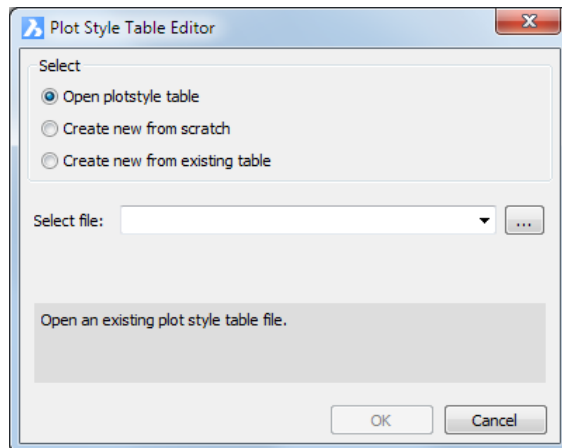


4. Do one of the following:
 - Select either *CTB* or *STB* from the *Table type* selection button.

CTB (Color-dependent Plot Style Table) - bases plots on entity colors. Plotting parameters, such as pen thickness and linetype, are matched to the color of entities. This system is older and less flexible than named plot styles, but is the default in all new drawings. Creates a CTB file.

STB (Named Plot Style Table) - bases plots on plot styles. Plotting parameters can be given to every entity and every layer. This system is new and more flexible, but may be incompatible with other software that reads DWG files. Creates an STB file.

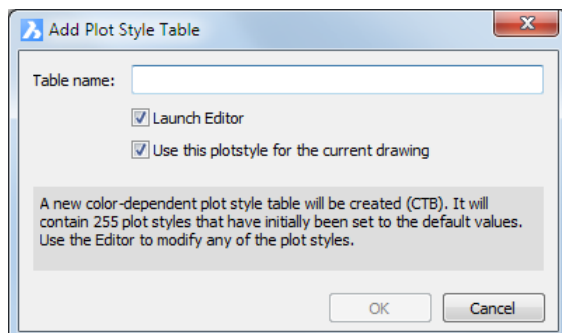
- Choose *Create from existing table*, then choose a file in the drop down list box or click the browse button to select an existing CTB or STB file.



5. Type a name in the *Table name* field, then click the *OK* button.
The Plot Style Table Editor dialog displays.

To create a new plot style table in the Print dialog

6. Open the Print dialog.
7. Click the *Create new plot style* button (📄✚) on the *Print* dialog.
The *Add Plot Style Table* dialog displays.




8. Type a name in the *Table name* field, then click the *OK* button.
If the *Launch Editor* option is chosen, the Plot Style Table Editor dialog displays.
If the *Use this plotstyle for the current drawing* option is chosen, the new plot style table is used in the current layout.

NOTE The above procedure creates a CTB file in CTB-type drawing and a STB file in a STB-type drawing.

To edit a plot style table

1. Open the Plot Styles Manager.
2. Double click the plot style table you want to edit.
The Plot Style Table Editor dialog displays.

To edit a plot style table in the Print dialog

1. Open the Print dialog.
2. Select a plot style table from the *Plot Style Table* list button.
3. Click the edit *Edit plot style* button ().
The Plot Style Table Editor dialog displays.

Batch Plot

Commands: PUBLISH, SHEETSET

The *Publish* command allows to print a sheet list (= a list of model space or paper space layouts) to a printer. A sheet list can be saved to a file.

The *SheetSet* command manages sheet sets and allows to print, publish or etransmit a sheet set as a whole or a selection of sheets.

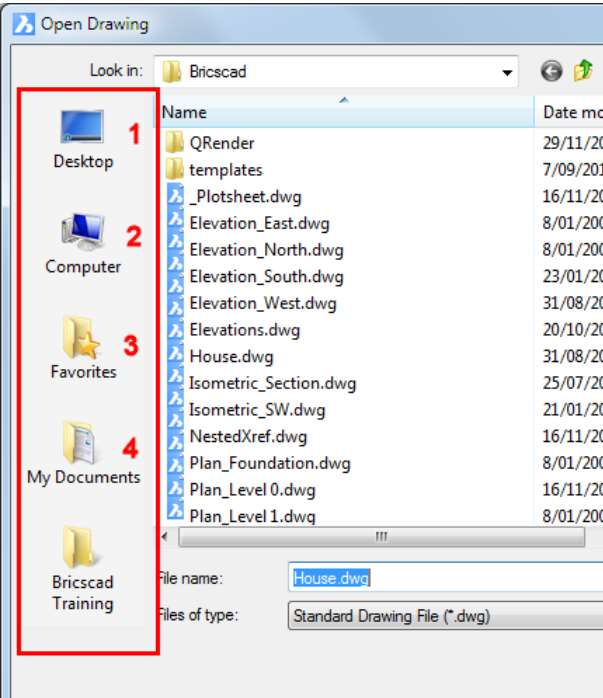
Managing Drawings

Opening and saving drawings

Commands: OPEN, SAVE, SAVEAS, SAVEALL

The way BricsCAD opens and saves drawings is controlled through a series of system variables and user preferences.

In the table below the SYSTEM VARIABLES are fully capitalized.

Name	Title	Description
ShowFullPathInTitle	Display full path in title	Controls whether the full path of a drawing displays in the title bar, or the file name only.
DrawingPath	Drawings path	Specifies an additional folder in the file dialog for the Open and SaveAs commands. On the Windows platform: the fifth folder in the <i>Open File</i> dialog's <i>Places</i> bar.
PlacesBarFolder1 - 4	First, Second, Third and Fourth folder	<p>Sets the folders 1 through 4 in the <i>Places</i> bar of the non-standard <i>Open File</i> dialog (Windows platform only). Values between 0 and 5 are accepted: 0=Desktop, 1=My Computer, 2= My Documents, 3= Favorites, 4=Network, 5=My Recent Documents.</p> <p>Tip: Place shortcuts to your favorite drawing folders on your desktop or in your Favorites folder.</p> 

THUMBSIZE	Thumbnail preview image size	<p>Range is 0 - 8; Default = 1</p> <p>0 (64 x 64)</p> <p>1 (128 x 128)</p> <p>2 (256 x 256)</p> <p>3 (512 x 512)</p> <p>4 (1024 x 1024)</p> <p>5 (1440 x 1440)</p> <p>6 (1600 x 1600)</p> <p>7 (1920 x 1920)</p> <p>8 (2560 x 2560)</p>
INDEXCTL	Index Control	<p>Determines whether layer and/or spatial indexes are created and saved.</p> <p>The layer index lists all entities in the drawing and what layer they are on. The spatial index stores the location of all the entities in 3D space. The indexes are used to load Xrefs if demand loading is enabled. If a layer index exists, entities on layers which are Off or Frozen are not loaded. If a spatial index exists, entities in clipped Xrefs which are outside the clipping boundary are not loaded. The demand loading of Xrefs is controlled through the XLOADCTL system variable).</p> <p>Creating indexes slows down the saving of a drawing, but increases the speed of Xrefs. Therefore create indexes only for drawings that will be used as an external reference.</p> <p>0 = No indexes, 1 = Layer index only, 2 = Spatial index only, 3 = Layer index and spatial index.</p>

XLOADCTL	Xref load control	<p>Controls demand loading of entities in external references.</p> <p>If demand loading is turned on you will experience a performance increase:</p> <p>When clipping external references which are saved with a spatial index (INDEXCTL = 2 or 3), entities in the external reference within the clipped area are loaded only. If the clipping area is modified, more or less entities are loaded from the reference drawing.</p> <p>When freezing layers in external references which are saved with a layer index (INDEXCTL = 1 or 3), entities in the external reference on thawed layers are loaded only. If xref-dependent layers are thawed more entities are loaded.</p> <p>0: Demand loading is OFF - The entire external reference is loaded.</p> <p>1: Demand loading is ON - Referenced drawings are kept open and locked. Other users can open such drawings only as read-only.</p> <p>2: Demand loading is ON - Temporary copies of referenced drawings are kept open and locked. Other users can open and modify the referenced drawings.</p>
ISAVEBAK	Incremental save backup	<p>Controls the creation of a backup file (BAK). Improves the speed of incremental saves if switched off, especially for large drawings.</p> <p>Please notice that in some cases (i.e. a power failure during a save), it might be possible drawing data are lost.</p>
ISAVEPERCENT	Save percent	<p>This system variable controls how often BricsCAD performs a full save. A full save reorganizes all the data in the drawing database and creates a compact file. A partial save stores all changes at the end of the drawing data base. A full save takes longer to be executed than a partial save. If ISAVEPERCENT = 0, each save is a full save. The higher the value of ISAVEPERCENT, the longer the time between two full saves.</p>
NFILELIST	Recent File List	<p>Specifies the number of files shown in the <i>Recent Files</i> flyout of the <i>File</i> menu. Values between 0 and 20 are accepted. Default = 10.</p>
PROXYGRAPHICS	Proxy graphics	<p>Determines whether images of proxy entities(1) are saved in the drawing. If switched off, a bounding box displays instead.</p>
PROXYNOTICE	Proxy notice	<p>Displays a notice when you open a drawing containing proxy entities(1) created by an application that is not present.</p>

PROXYSHOW	Proxy show	Controls how proxy entities(1) display in a drawing. 0 = Not displayed, 1 = Graphic images, 2 = Bounding box.
PROXYWEBSEARCH	Proxy web search	Determines whether BricsCAD checks for object enablers: 0 = Never, 1 = If a live Internet connection is present.
SaveFormat	Save format	Sets the default dwg version to save a drawing.
SAVETIME	Save time interval	Sets the interval in minutes for automatic saves. If set to zero, automatic saves are turned off. Automatic saves are created with a .SV\$ extension in the folder defined by the SAVEFILEPATH system variable.
WorspaceSecurity	Enable macros	Defines whether macros are enabled when loading a VBA-project.
AutosaveChecksOnlyFirstbitDBMOD	Ignore all but first bit of DBMOD(2) for autosave	If this user preference is switched on, no autosave file will be created for files that are viewed only.
CloseChecksOnlyFirstBitDBMOD	Ignore all but first bit of DBMOD(2) for close	If this user preference is switched on, you can view a file, then close it without getting a request to save the file.

(1) Proxy entities are custom entities which are created by an add-on application. If the add-on is currently not loaded or not installed, such entities cannot be displayed properly.

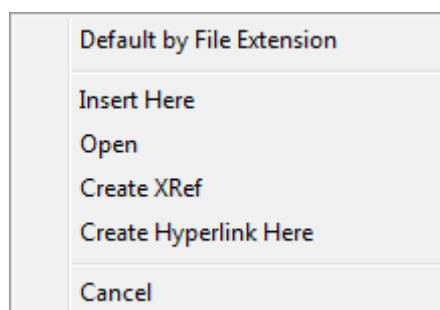
(2) DBMOD: Stores the status of the drawing modification as a bit-code as the sum of the following values: 1 = Object database modified, 4 = Database variable modified, 8 = Window modified, 16 = View modified, 32 = Field modified.

Using the Drag-and-Drop option menu

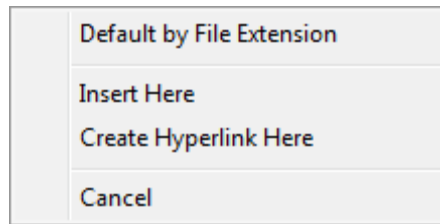
1. Select a file in a *Windows Explorer* dialog, then hold down the right mouse button and drag the file onto the drawing area of the BricsCAD application window.

A context menu displays

When a drawing file is selected:



When another file type is selected:



2. Do one of the following:
 - Choose *Default by File Extension* to
 - open a drawing file (*.DWG, *.DXF or *.DWT)
 - attach a PDF
 - attach a raster image file
 - insert a file as an OLE object (e.g. an Excel spreadsheet)
 - Choose *Insert Here* to:
 - insert a drawing file (*.DWG, *.DXF or *.DWT) as a block in the current drawing.
 - attach a PDF
 - attach a raster image file
 - insert the file as an OLE object
 - Choose *Open* to open a drawing file (*.DWG, *.DXF or *.DWT).
 - Choose *Create Xref* to attach a drawing file (*.DWG, *.DXF or *.DWT) as an external reference to the current drawing.
 - Choose *Create Hyperlink Here* to create an hyperlink to the selected file.

Getting drawing information

You can retrieve two types of information from a drawing:

General information about the drawing.

Information about the entities in the drawing

General drawing information

Commands: DWGPROPS, SETVAR, STATUS, TIME

The *Dwgprops* command opens the Drawing Properties dialog box, showing the general information and user defined properties stored with a drawing.

The *Setvar* command displays or changes the values of system variables (short for "set variables").

The *Status* command reports the status of the drawing: drawing name, the status of the most important system variables and the total number of entities in the drawing.

The *Time* command reports on time in the drawing: current time, creation time, last update, total editing time and user elapsed time.

Information about entities

Commands: STATUS, AREA, MASSPROP, ID, LIST

The *Status* command reports the total number of entities in the drawing.

Using Quick Select you can see the number of entities for each category.

The *Area* command finds the area and perimeter (or length) of 2D entities.

The *Massprop* command reports the area, perimeter, and other mathematical properties of 3D solids and 2D regions (short for "mass properties").

The *ID* command reports the x,y,z coordinates of picked points.

The *List* command lists the properties of selected entities.

Creating a drawing

Commands: NEW, NEWWIZ and QNEW

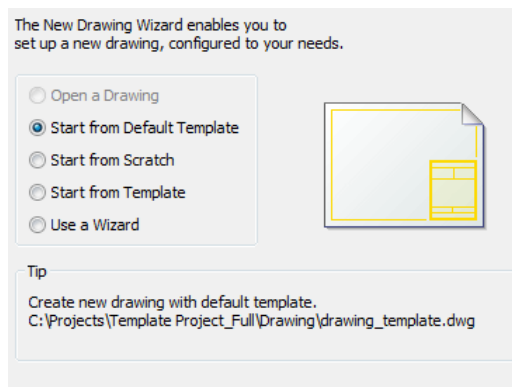
In BricsCAD you can create new drawings:

- Using a default template drawing.
- Starting from scratch.
- Using a template drawing of your choice.
- Using a wizard.
- Double clicking a *.dwt file in a *Windows Explorer* window.

To create a drawing using a default template drawing

First method:

1. Do one of the following:
 - Choose *New wizard...* in the *File* menu.
 - Type *newwiz* in the command window, then press Enter.
2. On the *Create New Drawing* dialog choose *Start from Default template*, then click the *Next* button.
The drawing is created as a copy of the default template.



Second method:

Do one of the following:

- Click the *QNew* tool button (📄➕) on the *Standard* toolbar.
- type *qnew* in the command bar, then press Enter.

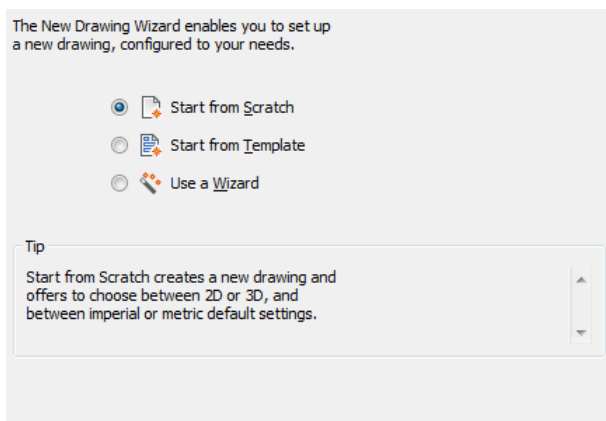
The drawing is created as a copy of the default template defined by the *BASEFILE* system variable. If the default template drawing is not specified, the *Create New Drawing* wizard is launched if the *STARTUP* system variable is *ON*, otherwise the *Select Template* dialog displays.

To set the default template drawing

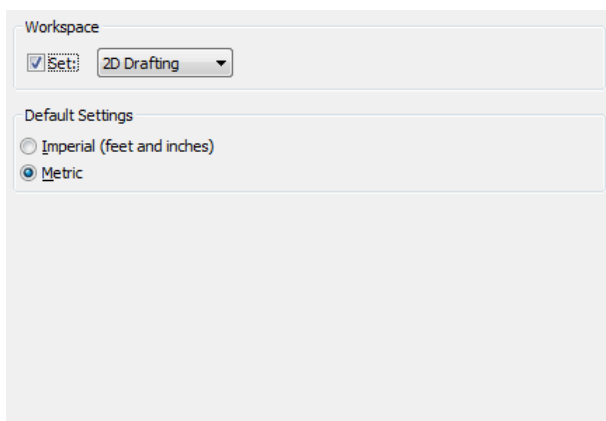
1. Open the Settings dialog.
2. Under *Program Options / Files / Templates* select *Template (BASEFILE)*
3. Click the *Browse* button.
4. On the *Choose a File* dialog, select a drawing file (*.dwg) or drawing template (*.dwt), then click the *Open* button.

To create a drawing from scratch

1. Do one of the following:
 - Choose *New wizard...* in the *File* menu.
 - Type *newwiz* in the command window, then press Enter.
2. On the *Create New Drawing* dialog choose *Start from Scratch*, then click the *Next* button.



3. (option) Check the *Set Workspace* option, then choose a workspace from the list button.
See *Workspaces* to learn more about workspaces.
4. Choose between either *Imperial* or *Metric* default settings.



5. Click the *Finish* button to create the drawing.

NOTE Using the above procedure it depends on the status of the *BASEFILE* and *PSTYLEPOLICY* variables whether *CTB* (Color table) or *STB* (Named plot style table) will be used to set up plotter configurations in the newly created drawing.

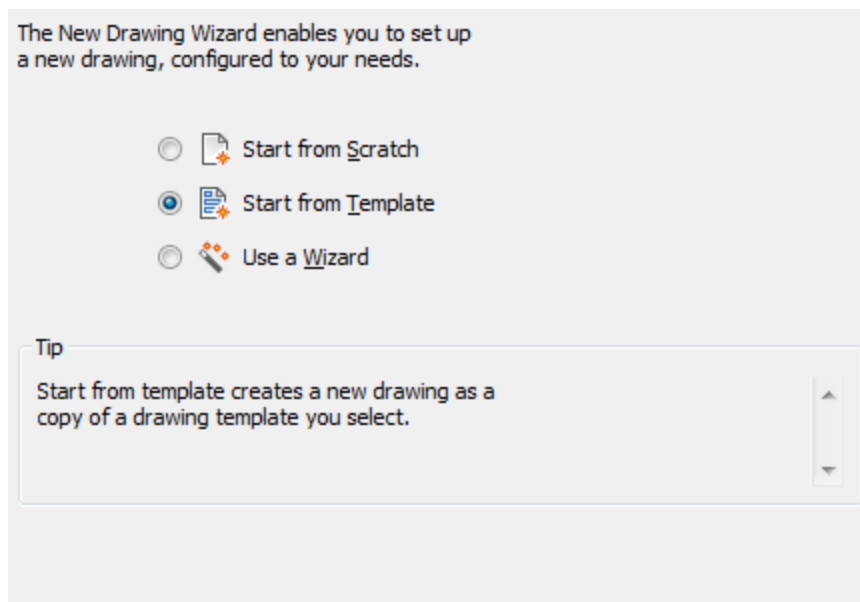
BASEFILE	PSTYLEPOLICY	CTB or STB?
not specified	OFF	STB
not specified	ON	CTB
specified	ON or OFF	as defined in the template drawing

To set the Plot style policy

1. Open the Settings dialog.
2. Under *Program Options / Plot and publish* select *Plot style policy (PSTYLEPOLICY)*.
3. Do one of the following:
 - Choose *Associate an entity's plot style with its color* for *Color dependent* plot styles.
 - Choose *No association between color and plot style* for *Named* plot styles.

To create a drawing using a template drawing

1. Do one of the following:
 - Choose *New wizard...* in the *File* menu.
 - Type *newwiz* in the command window, then press Enter.
2. On the *Create New Drawing* dialog choose *Start from template*, then click the *Next* button.



3. Do one of the following:
 - Choose a template drawing in the *Select a template list*.
By default this is the content of the *Template* folder in the Local root folder.
Edit the *Template Path* variable to set your template folder differently.
 - Click the *Browse...* button to select a template drawing.
You can choose between **.dwg* and **.dwt* file types.
- The drawing is created as a copy of the selected template.

To create a drawing using the New command

1. Do one of the following:
 - Choose *New...* in the *File menu*.
 - Type *new* in the command bar, then press Enter.
- The *Select template* dialog opens showing the content of the *Template* folder as defined in the *Template Path* variable.

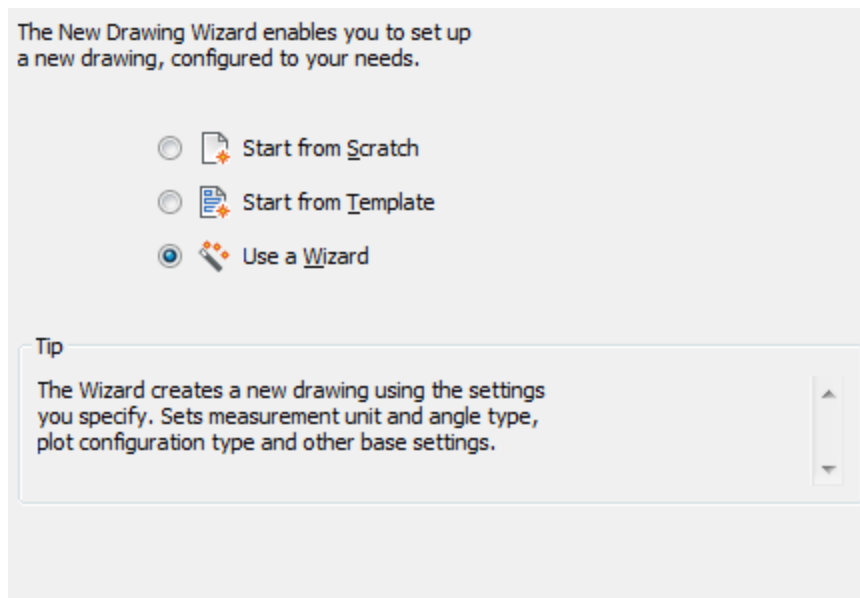
2. Select the template drawing, then click the *Open* button.
The drawing is created as a copy of the selected template.

To set the Template Path variable

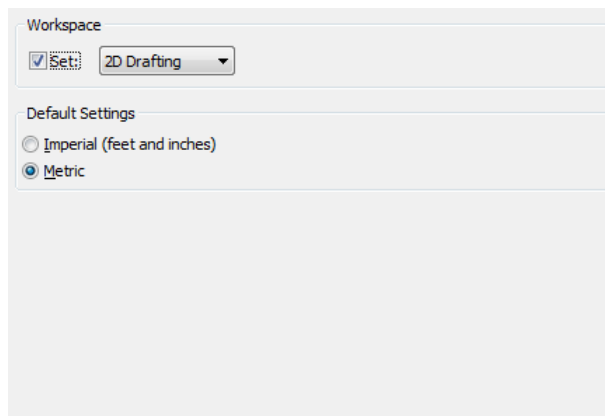
1. Open the Settings dialog.
2. Under *Program Options / Files / Templates* select *Template Path (templatePath)*
3. Click the *Browse* button.
4. On the *Browse for folder* dialog, select a folder , then click the *OK* button.

To create a drawing using a wizard

1. Do one of the following:
 - Choose *New wizard...* in the *File* menu.
 - Type *newwiz* in the command window, then press Enter.
2. On the *Create New Drawing* dialog choose *Use a Wizard*, then click the *Next* button.



3. (option) Check the *Set Workspace* option, then choose a workspace from the list button.
See *Workspaces* to learn more about workspaces.
4. Choose between either *Imperial* or *Metric* default settings, then click the *Next* button.



5. Select the *Unit of measurement* and the display *Precision*, then click the *Next* button.

Choose any one of the linear units of measurement listed below as the default for your drawing.

Unit of measurement

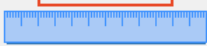
☐ Scientific

☒ Decimal

☐ Engineering

☐ Architectural

☐ Fractional



Precision: 0.0000

6. Select the *Angle of measurement* and the display *Precision*, then click the *Next* button.

Choose any one of the angular units of measurement listed below as the default for your drawing.

Angle of measurement

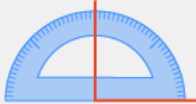
☒ Decimal Degrees

☐ Degrees/Minutes/Sec

☐ Grads

☐ Radians

☐ Surveyor's units



Precision: 0.00


7. Choose between *Color Dependent* plot styles (CTB) or *Named* plot styles (STB), then click the *Next* button.

Do you want to use Color Tables (CTB) or Style Tables (STB) to set up plotting configurations?
Color Tables assign plot settings based on the color of entities while Style Tables offer a more advanced and complex way to control plotting configurations.

Plot Style

☒ Color Dependent (CTB)

☐ Named (STB)




8. Select the default *Color* and *Linetype*.

Choose how you want to create entities in your new drawing.

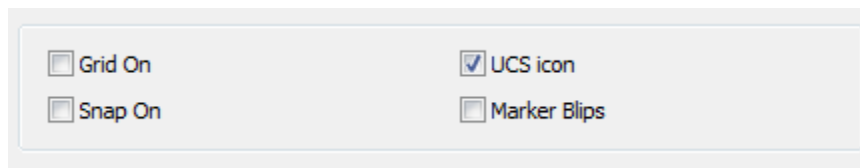
Once you are working in the drawing, you will be able to change any of these as needed.

Entity

Color:  ByLayer

Linetype: ByLayer

9. Define the *Grid*, *Snap*, *UCS icon* and *Marker Blips* settings.



10. Click the *Finish* button to create the drawing.
 11. (option) Choose *Save As...* in the *File* menu to save the drawing as a template in the *Templates* subfolder of the Local root folder.

Drawing Explorer

Commands: EXPLORER, LAYER, LAYERSTATE, LINETYPE, MLSTYLE, MLEADERSTYLE, STYLE, DIMSTYLE, TABLESTYLE, EXPUCS, VIEW, VISUALSTYLES, LIGHTLIST, MATERIALS, RENDERPRESETS, EXPBLOCKS, XREF, IMAGE, PDF, eTRANSMIT, PAGESETUP, SECTIONPLANESETTINGS



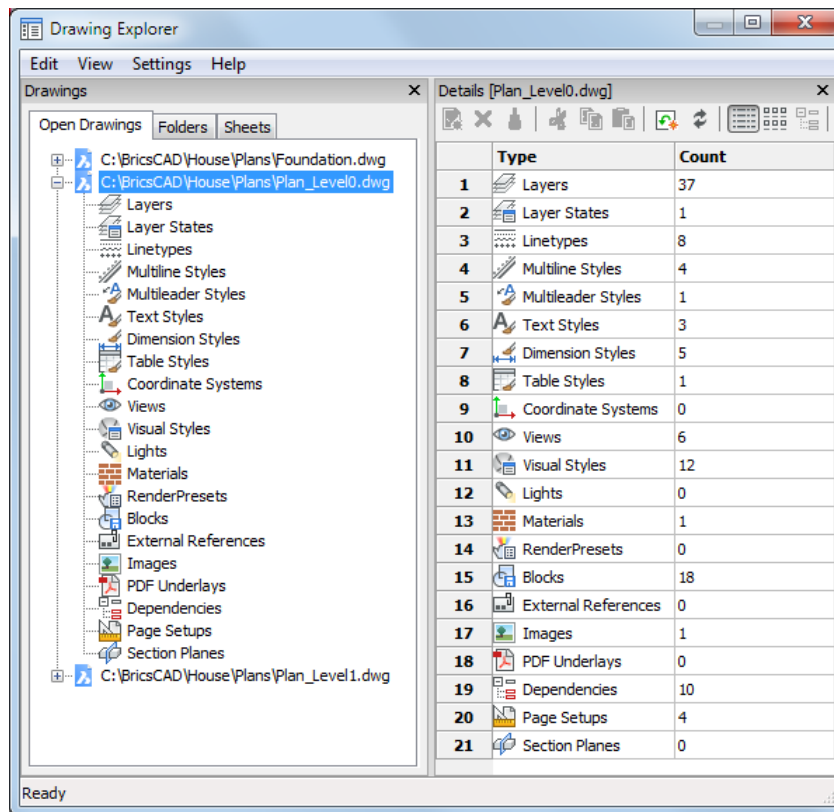
The Drawing Explorer (Internet connection needed)

The *Drawing Explorer* window consists a number of sub-windows or panes:

- *Drawings*:
- *Open Drawings*: a list of all drawings that are currently open.
- *Folders*: your favorite drawing folders
- *Details*: the details of the selected drawing or the details of the selected category in a drawing, e.g. layers, blocks, images, ...
- *Preview*: a preview of the selected drawing, layer, block, view, dimension style, table style, XRef, image or material.
Use the arrow keys to browse through the items.
- *Editor*: to define or edit the properties of *Multiline Styles*, *Multileader Styles*, *Table Styles*, *Views*, *Visual Styles*, *Sunlight settings*, *Materials*, *Render Presets* and *Section Planes*.


You can select which panes you want to have open or closed in the View menu.

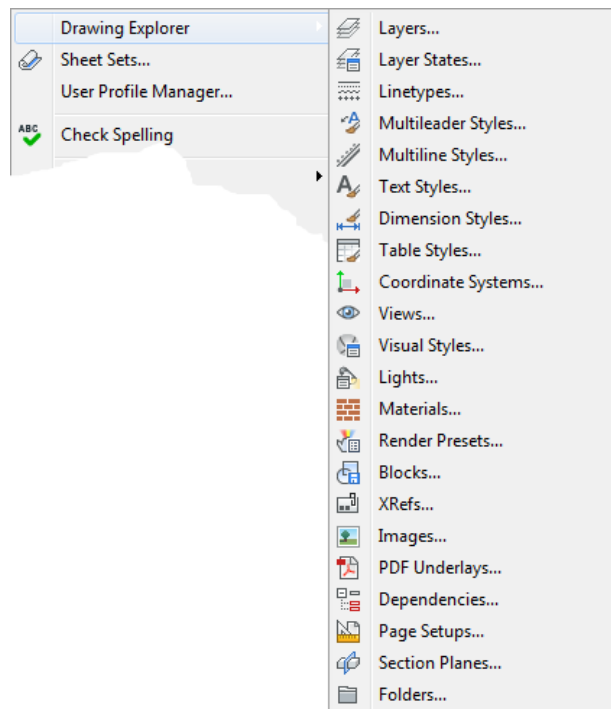
BricsCAD remembers the display settings for each category.



To open the Drawing Explorer

To open the *Drawing Explorer* window do one of the following:

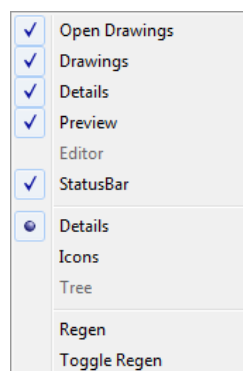
- Click the *Drawing Explorer* tool button () on the *Standard* toolbar. The *Drawing Explorer* window opens showing the details of the most recently used category.
- Click the *Layer*, *Linetype*, *Text Style* or *Dimension Style* field in the *Status Bar*. The *Drawing Explorer* window opens showing the details of the clicked definitions category in the current drawing. (See Customizing the Status Bar to enable/disable fields in the *Status Bar*.)
- Choose a *Drawing Explorer* in the *Tools* menu, then select a category in the flyout. The *Drawing Explorer* window opens showing the details of the selected category.



- Type *explorer* in the command bar, then press Enter. The *Drawing Explorer* window opens showing the details of the most recently used category.

To control the Drawing Explorer layout

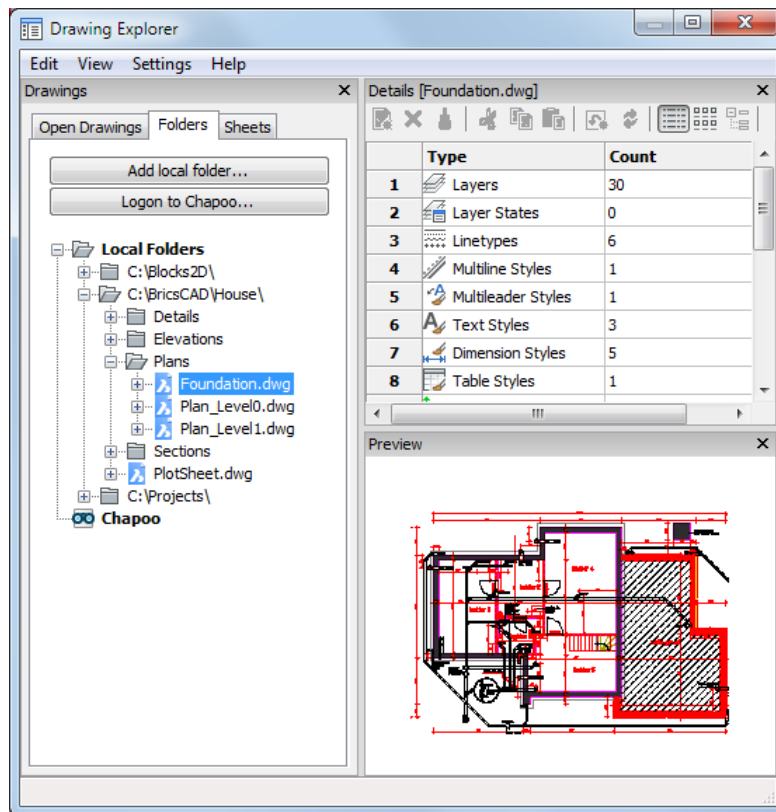
- In the *View* menu, check/uncheck the sub-window you want to open or close.



- If you click a settings category in the *Drawings* sub-window when the *Details* sub-window is closed, the *Details* window is re-opened, showing the content of the selected settings category.
- To close a sub-window, click the close button (X) in its title bar.
- To restore the default layout choose *Restore Default Explorer Layout* in the *Settings* menu.

Opening a drawing

1. To open a drawing using the Drawing Explorer do the following
2. Launch the *Drawing Explorer*.
3. In the *Drawings > Folders* sub-window browse to the folder of the drawing.



4. Double click the drawing.
The drawing opens, while the *Drawing Explorer* window stays open.
5. (optional) Repeat steps 2 and 3 to open more drawings.
6. Close the *Drawing Explorer*.

NOTE When you select a drawing, a preview displays in the *Preview* sub-window.

Adding a drawing folder

1. In the *Drawings > Folders* sub-window of the *Drawing Explorer* main window click the *Add local folder...* button.

The *Browse For Folder* window opens.


2. In the *Browse For Folder* window do one of the following:
 - To add an existing folder: select the folder you want to add.
 - To create a new folder: click the *Make New folder* button.
3. Click the *OK* button.
The folder is added.

NOTE Drawings in the subfolders of the selected folder can be opened too.
The parent folder(s) of the selected folder is (are) greyed out in the drawing folder tree.
Drawings in such folders cannot be opened.

To insert a block from an unopened drawing

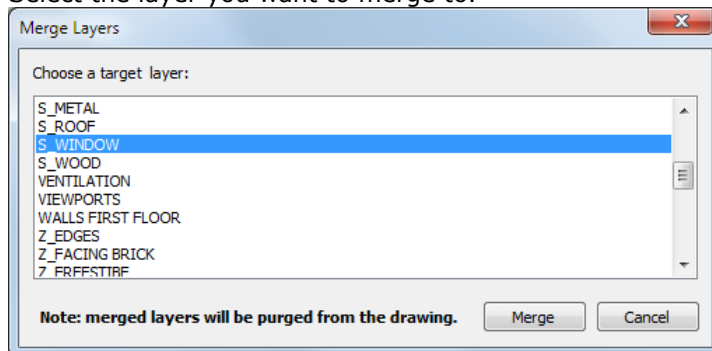
The Block Manager feature of the *Drawing Explorer* allows to insert blocks in the current drawing from unopened drawings.

To copy definitions between drawings

1. Open both the source and the target drawing.
2. Launch the Drawing Explorer.
3. In the source drawing select the category you want to copy items from, e.g. *Blocks*.
4. (option) In the *Details* toolbar, click the *Icon View* button .
5. Select the item(s) you want to copy, e.g. a series of blocks.
Press and hold the Ctrl key to select multiple items.
6. Press and hold the left mouse button to drag the selection set to the *Open Drawings* section of the *Drawing Explorer*.
7. Release the left mouse button when on the target drawing name.
The selected items are copied to the target drawing.
You will be prompted to overwrite items of the same name.

To merge the content of two layers

1. Open the Drawing Explorer > Layers dialog window.
2. Select the layer you want to merge into another layer.
3. Choose *Merge to...* in the *Edit* menu.
The *Merge Layers* dialog box displays.
4. Select the layer you want to merge to.



5. Click the *Merge* button.
The content of the first layer is merged to the second layer.
The first layer is purged from the drawing.

Repairing a drawing

Commands: AUDIT and RECOVER

The *Audit* command analyzes the integrity of the current drawing and optionally tries to fix errors.

The *Recover* command repairs a damaged drawing, then opens the drawing.

To analyze the integrity of the current drawing

1. Do one of the following:
 - Choose *Audit* in the *File | Drawing Utilities* menu.
 - Type *audit* in the command bar, then press Enter.

The command bar reads: Fix any errors detected? Yes/<No>

2. Do one of the following:
 - Choose *Yes* in the context menu or type *Y*, then press Enter to analyze the integrity of the current drawing and fix the errors detected.
 - Choose *No* in the context menu or type *N*, then press Enter to analyze the integrity of the current drawing.

The drawing is analyzed.

3. (option) Press F2 to open the *Prompt History* window in order to read the audit report:

objects audited

Total errors found during audit #, fixed #.

NOTE Set the *AUDITCTL* settings variable to 1, to let *Audit* create an ASCII file describing problems and the action taken. This report, with the file extension .adt, is placed in the same directory as the current drawing .

Use the *Recover* command to repair the errors that *Audit* cannot fix.

To repair a drawing

1. Do one of the following:
 - Choose *Recover* in the *File | Drawing Utilities* menu.
 - Type *recover* in the command bar, then press Enter.

The *Open Drawing* dialog window displays.
2. Select the drawing, then press the *Open* button on the *Open Drawing* dialog window.

The drawing is repaired and - if the repair succeeded - opened.
3. (option) Press F2 to open the *Prompt History* window in order to read the recover report.

Drawing Properties

Command: DWGPROPS

The *Dwgprops* command opens the *Drawing Properties* dialog box, showing the general information and user defined properties stored with a drawing.

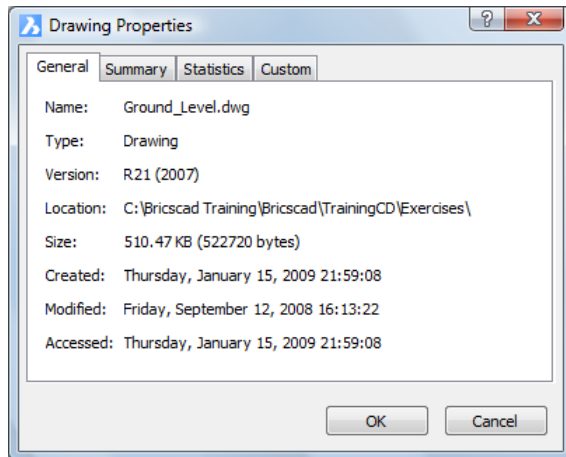
Drawing properties include: general information, summary, statistics and user defined custom properties.

Drawing properties can be referenced in a field in multiline text entities or tables.

To open the Drawing Properties dialog

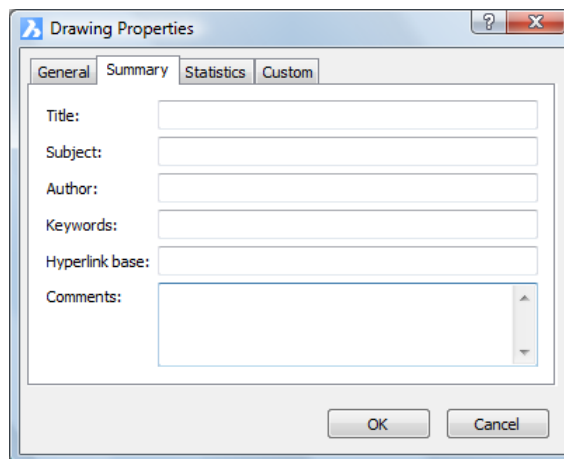
do one of the following:

- Choose *Drawing Properties...* in the *File* menu.
- Type *dwgprops* in the command bar and press Enter.



Defining default properties

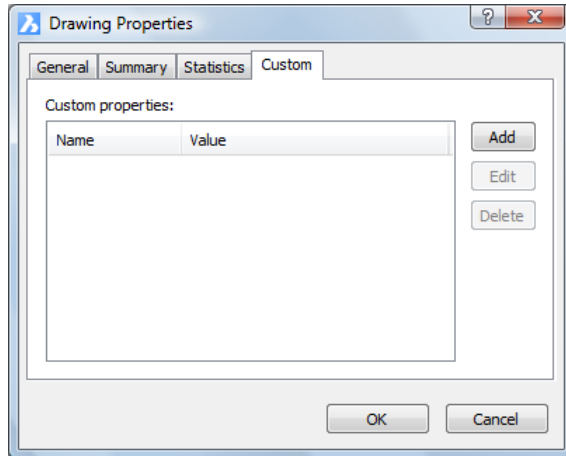
1. Open the Drawing Properties dialog.
2. Open the *Summary* tab page.



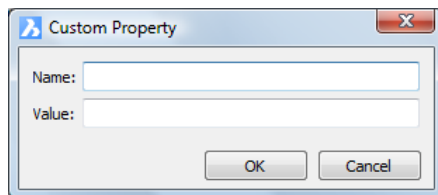
3. Fill out the property fields
4. Click the *OK* button.

Adding custom properties

1. Open the Drawing Properties dialog.
2. Open the *Custom* tab page.



3. Click the *Add* button.
The *Custom Property* dialog displays.



4. Type a name in the *Name* field.
The custom property name is listed under *Document* in the Field dialog.
5. Type a value in the *Value* field.
The value displays in multiline texts where the custom property is inserted as a field.

Drawing Security Options

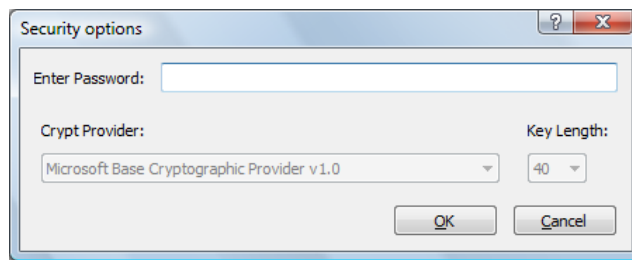
Command: SECURITYOPTIONS

The *Securityoptions* command allows to protect your drawings with a password. Password protected drawings cannot be opened without entering the password first.

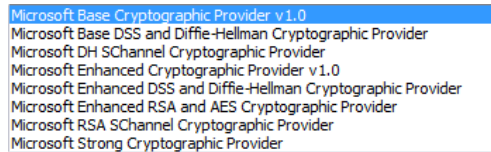
To set the password

1. Do one of the following:
 - Choose *Security Options...* in the *File | Drawing Utilities* menu.
 - Type *securityoptions* in the command bar, then press Enter.

The *Security options* dialog opens:



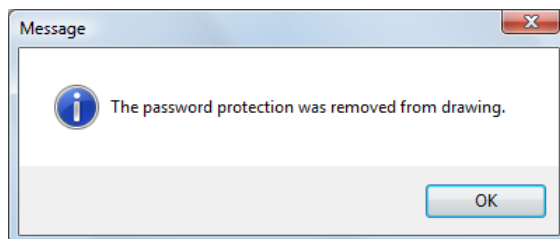
2. Type the password in the *Enter Password* field.
3. Choose a crypt provider from the *Crypt Provider* list.



4. Set the *Key Length* to 40, 48 or 56.
5. Click the *OK* button or press Enter.
The *Security options* dialog closes.
The *Password* dialog opens.
6. Type the password in the *Confirm Password* field.
7. Click the *OK* button or press Enter.
The password is defined.

To remove the password

1. Type *securityoptions* in the command bar, then press Enter.
The *Security options* dialog opens.
2. Empty the *Enter Password* field.
3. Click the *OK* button or press Enter.
A message box displays to confirm that the password is removed.



4. Click the *OK* button or press Enter to close the message box.

Export to PDF

Command: EXPORT

Drawing files can be saved as a PDF file, optionally including layers.

Export to PDF settings

Choose *PDF Export Options...* in the *File* menu to display the *PDF Export* settings in the *Settings* dialog.

PDF export	
Pdf embedded fonts	<input checked="" type="checkbox"/> Enable embedded TTF fonts
Pdf TTF text as geometry	<input type="checkbox"/> Convert TTF text to geometry
Pdf SHX text as geometry	<input type="checkbox"/> Convert SHX text to geometry
Pdf simple geometry optimization	<input checked="" type="checkbox"/> Enable simple geometry optimization
Pdf zoom to extents mode	<input checked="" type="checkbox"/> Zoom to extents
Pdf layer support	[1] Use layers/Don't include OFF-layers
Pdf layouts to export	[0] Active
Pdf papersize override	<input type="checkbox"/> Enable papersize override
Pdf overridden paper width	210
Pdf overridden paper height	297
Pdf use plotstyles	<input checked="" type="checkbox"/> Enables usage of plotstyles
Hatch to bitmap DPI	300

Pdf embedded fonts: Controls whether True Type Fonts are embedded in the PDF export.

Pdf TTF text as geometry: If selected, TTF text is exploded to geometry.

Pdf SHX text as geometry: If selected, SHX text is exploded to geometry.

Pdf simple geometry optimization: Enable simple geometry optimization: e.g. separate segments to one polyline, use of Bezier curve control points.

Pdf zoom to extents mode: If exporting model space, zoom to extents first, if the current model space view is zoomed in.

Pdf layer support: Controls whether layers are included in the PDF export.
The options are: do not export layers, export layers which are on only, export all layers.

Pdf layouts to export: Controls which layouts are exported.

The options are:

Active: only the active layout or model space is exported as a single-page pdf.

All: All layouts, including model space are exported as a multi-page pdf.

Pdf papersize override: If on, the papersize as defined in the BricsCAD Print settings is overridden. The papersize width and height defined by *PdfPaperWidth* and *PdfPaperHeight* is used instead.

Pdf overridden paper width: Width (in mm) of the PDF paper, if *PdfPaperSizeOverride* is on.

Pdf overridden paper height: Height (in mm) of the PDF paper, if *PdfPaperSizeOverride* is on.

Pdf use plotstyles: If on, the plotstyle of the layout controls the color and lineweight in the PDF export.

Hatch to bitmap DPI: Defines the resolution of hatch patterns and gradient fills in DPI (= dots per inch).

Exporting a drawing to PDF

1. (option) Adjust the Export to PDF settings.
2. Choose *Export...* in the *File* menu.
The *Export Drawing As* dialog opens.
3. In the *Save as type*: list select *Adobe PDF (*.pdf)*.
4. Select a folder to save the PDF file.
5. (option) Type a name in the *File name* field to replace the default <DRAWING_NAME>.pdf name.
6. Click the *Save* button to create the PDF.

Etransmit

Command: ETRANSMIT

The *Etransmit* command creates a package of a drawing file and all its dependencies, such as: external references, images, font files, plot configuration files, plot style tables and font map files.

The root drawing and all its dependencies can be copied to a folder are compressed in a ZIP file.

Optionally you can send the ZIP file attached to an email.

Digitizing Tablet

Command: TABLET

The Tablet command configures and calibrates a digitizing tablet and toggles tablet mode.

This command requires that Wintab32.dll from Wacom Technology, Corp. be installed on the computer. Download Wintab32.dll from the internet, then place the file in your system directory: C:\Windows\System32.

A digitizing tablet, also called a graphics tablet, a graphics pad or drawing tablet, is a computer input device that allows you to hand-draw images and graphics, similar to the way one draws images with a pencil and paper. In a CAD program a digitizing tablet can be used to trace paper drawings into a drawing file or to launch drawing commands from a digitizing tablet overlay.

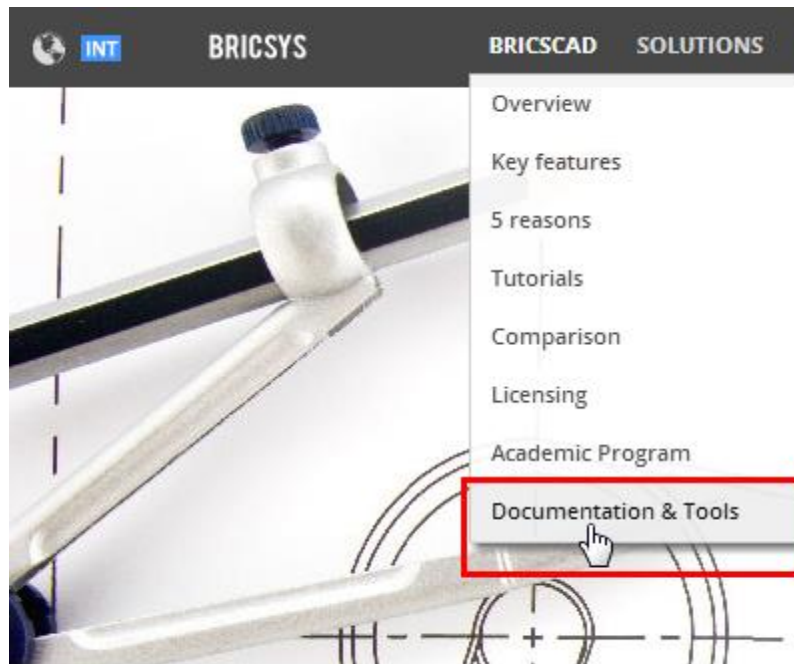
To use a digitizing tablet in BricsCAD

In order to prepare BricsCAD for using a digitizing tablet do the following:

- Install the appropriate driver for your tablet.
- Make sure Wintab32.dll exists in C:\Windows\System32.
- Download the tablet overlay and CUI files.
- Print the BricsCAD tablet overlay.
- Load the overlay CUI file.
- Start BricsCAD, then initialize the tablet.
- Configure the tablet: part of the tablet surface is used as the menu area, another part as the screen pointing area.
- Calibrate the tablet: a calibrated tablet can be used to trace the geometry of a paper drawing or image into a drawing.

To download the tablet overlay and CUI files

1. Go to the *BricsCAD Documentation and Tools* page on the *Bricsys* website.



2. Scroll down to the bottom of the page and click the *DOWNLOAD* button under *Tablet overlay*.

Tablet overlay

Publisher Bricsys

Overlay and CUI files to use your tablet with
BricsCAD.



3. Save the *tablet.zip* file.

The zip-file contains the following files:

- *overlay.png*: an image of the tablet overlay
- *overlay(A3).pdf*: to print the tablet overlay on a A3 paper size.
- *overlay(cm).dwg*: a DWG in which the *overlay.png* is attached as an image. The drawing contains two layouts to print the overlay using decimal units.
- *overlay(inch).dwg*: a DWG in which the *overlay.png* is attached as an image. The drawing contains two layouts to print the overlay using imperial units.
- *tablet.cui*: a partial cui file to be used with a tablet.
- *tablet(acadLike).cui*: an AutoCAD-like partial cui file to be used with a tablet.

Please make sure the *overlay.png* image file is extracted to the same folder as the DWG's.

To load the overlay CUI file

1. Extract the overlay CUI files to the *Support* folder of the *roamable root* folder (see BricsCAD User Files).
E.g. C:\Documents and Settings\<user name>\Application Data\Bricsys\BricsCAD\V...\en_US\Support on Windows XP;
C:\Users\<user name>\AppData\Roaming\Bricsys\BricsCAD\V...\en_US\Support on Windows Vista and Windows 7.
2. Load *Tablet.cui* as a partial CUI (see To load a partial CUI file).
The next time you start BricsCAD, *Tablet.cui* will be loaded automatically.

NOTE It might be necessary to repeat the above procedure after upgrading BricsCAD.

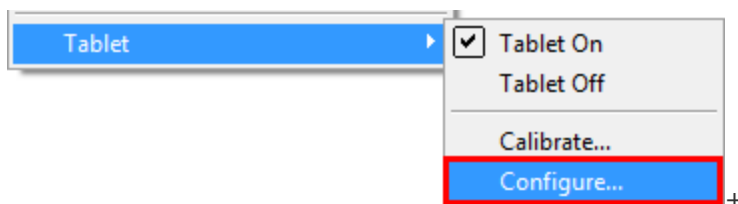
To switch the tablet ON / OFF

Do one of the following:

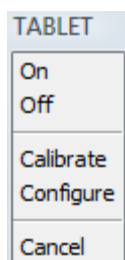
- Click the *TABLET* field in the Status Bar.
- Press the *F4* function key to toggle the tablet mode.
- Choose *Tablet On* or *Tablet Off* in the *Settings* menu.
- Type *tablet* in the command bar, press Enter, then type *ON* or *OFF* ;
or
click *ON* or *OFF* in the prompt box.

To Configure the tablet

1. Do one of the following:
 - Choose *Tablet* in the *Settings* menu, then choose *Configure...* in the flyout menu.



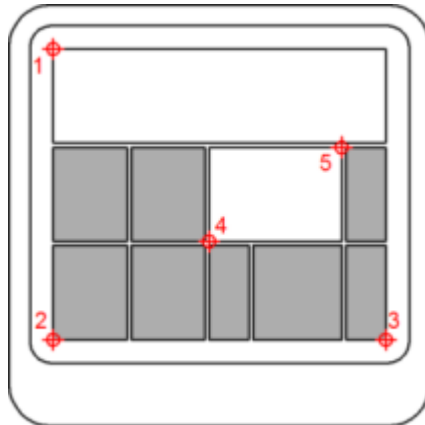
- Type *tablet* in the command bar and press Enter.
The command bar reads: *Tablet: ON/OFF/CALibrate/ConFiGure/<On>*;
a prompt menu displays:



Type *cfg* in the command bar and press Enter or choose *Configure* in the prompt menu.

The command bar reads: Do you want to align the tablet overlay? Yes/No/<No>:

2. Type *Y* in the command bar and press Enter or choose *Yes* in the prompt menu.
The command bar reads: Digitize upper left corner of the overlay:
3. Click on the mark in the upper left corner (1) of the digitizing tablet overlay.



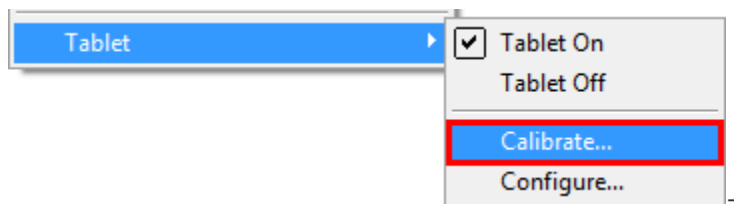
- The command bar reads: Digitize the lower left corner of the overlay.
4. Click on the mark in the lower left corner (2) of the digitizing tablet overlay.
The command bar reads: Digitize the lower right corner of the overlay.
 5. Click on the mark in the lower right corner (3) of the digitizing tablet overlay.
The command bar reads: Digitize the lower left corner of the screen pointing area:
 6. Click on the mark at position (4) on the of the digitizing tablet overlay.
The command bar reads: Digitize the upper right corner of the screen pointing area:
 7. Click on the mark at position (5) on the of the digitizing tablet overlay.
The command bar reads: Tablet configured.

•

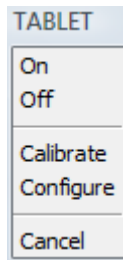
To calibrate the tablet

To calibrate the digitizing tablet you must specify at least two points. The more additional points you specify, the more accurate the digitizing process. Additional points are particularly recommended when digitizing a non-orthogonal image, such as an aerial photograph.

1. Do one of the following:
 - Choose *Tablet* in the *Settings* menu, then choose *Calibrate...* in the flyout menu.



- Type *tablet* in the command bar and press Enter.
The command bar reads: Tablet: ON/OFF/CALibrate/ConFiGure/<On>:
a prompt menu displays:



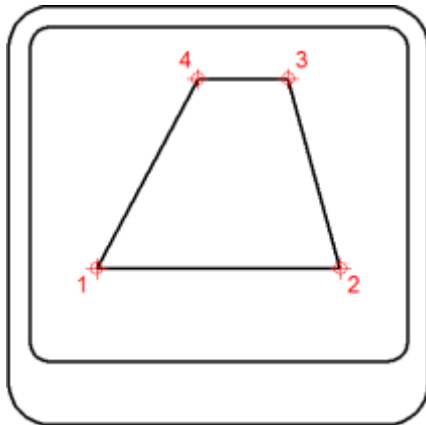
Type *cal* in the command bar and press Enter or choose *Calibrate* in the prompt menu.

The command bar reads: Digitize point #1:

2. Click a point on the digitizing tablet .
The command bar reads: Enter coordinates for point #1:
3. Do one of the following:
 - Click a point in the BricsCAD drawing window.
 - Enter the coordinates (x,y) in the command bar.

The command bar reads: Digitize point #2:

4. Repeat step 3 to specify the following calibration point.
The command bar reads: Digitize point #3 (or ENTER to end):
5. Do one of the following:
 - Repeat step 4 to specify an additional calibration point.
(You can specify up to 10 calibration points).
 - Press Enter to conclude the calibration process.
The command bar reads:
if 2 calibration points are specified: Tablet calibrated.
if 3 or more than 4 calibration points are specified: Select transformation type
Orthogonal/Affine:
if 4 calibration points are specified: Select transformation type
Orthogonal/Affine/Projective:



6. Select the transformation type.
Transformation refers to the calculation of the points on the screen that correspond to points you digitize on the tablet.



Orthogonal (left): To be used with accurate paper drawings. Orthogonal transformation maintains all angles and preserves relative distances. If only two calibration points are specified orthogonal transformation is generated automatically.

Affine (middle): Maintains parallel lines, but not necessarily the angles between intersecting lines.

Projective (right): Does not maintain parallel lines, nor angles.

Rendering

Commands: RENDER, SHADEMODE, RENDERPRESETS

Rendering is the process of generating an image from a 3D model. The model contains the geometry, viewpoint, texture, lighting and shading information. The result is a digital image which can be saved as a raster image file: bmp, jpeg or png.

Rendering in BricsCAD is based on the Redway3d rendering engine. By default the rendering engine will try to leverage the hardware on the graphics card. If experiencing problems, please switch off the *RenderUsingHardware* user preference. It is recommended to install the most recent drivers for your graphics card. The *IsRedSdkHardwareRenderingSupported* user preference reports if Red SDK hardware rendering is fully supported and the initial value of *RenderUsingHardware* user preference is set accordingly.

The *RenderPresets* command allows to create and edit render presets, and to set the current render preset through a dialog box.

Setting the render preferences

1. Open the Settings dialog.
2. Under *Program Options* expand the *Rendering* settings category.

☐ Rendering	
Current material	ByLayer
Lighting units	[2] Use International lighting units
Render using hardware	[1] Prefer hardware (faster)
Real world scale	<input checked="" type="checkbox"/> Render real-world scale materials
Sky status	[0] No sky
Texture map path	C:\Program Files\Bricsys\BricsCAD V14 en_US\Textures\1\
Tile mode light synch	<input checked="" type="checkbox"/> Synchronize lighting

Current material: Specifies the material of new entities.

Lighting units: Web lights can be created only if the LIGHTINGUNITS system variable is set to 1 (American lighting units) or 2 (International lighting units). (Not implemented yet)

Render using hardware: Choose *User software only (slower)* if hardware rendering fails.















Real world scale: Controls the rendering of materials with units set to real-world scale

Sky status: Specifies whether sky illumination is computed at render time. (Not implemented yet)

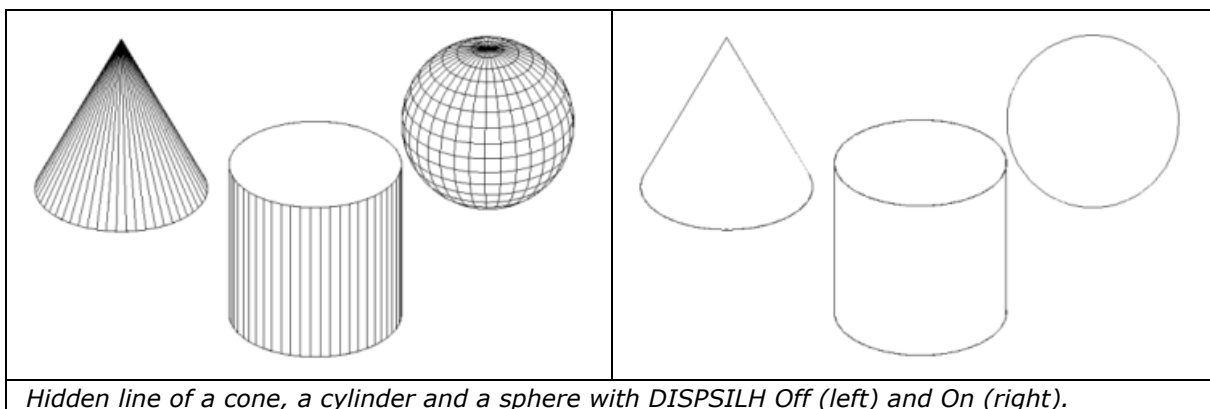
Texture map path: The TextureMapPath user preference defines the search path for texture map images. In the BricsCAD program folder exist three subfolders under *Textures*, each containing a number of texture files of the same name. Images in folder 1 have a size of 256 x 256 pixels, folder 2 contains images of 512 x 512 pixels, images in folder 3 have a size of 1024 x 1024 pixels. If the *Diffuse map* setting of a material uses the image name only (not path), you can control the quality of a rendered image by setting the *TextureMapPath* user preference to folder 1, 2 or 3.

Tile mode light synch: Controls the synchronization of lighting in all model space viewports. (Internal use only)

Rendering tools

Icon	Tool Name	Keyboard	Description
	Render	render	Photorealistic rendering using materials and lights
	Hide	hide	Hidden line view (*). If the previous shademode option was <i>2Dwireframe</i> , wireframe representation is restored if you zoom or pan. If the previous shademode option was other than <i>2Dwireframe</i> , the hidden line representation is kept until a different shade mode is chosen.
	Shade	shade	Shades the entities between the polygon faces. Curved surfaces are segmented.
	2D Wireframe	shademode + 2	Wireframe representation
	3D Wireframe	shademode + 3	Wireframe representation
	Hidden	shademode + H	Hidden line view (*)
	Flat	shademode + F	Shades the entities between the polygon faces. Curved surfaces are segmented
	Gouraud	shademode + G	Shades the entities between the polygon faces. Curved surfaces are smoother and more realistic
	Flat + Edges	shademode + L	Combines flat shading and wireframe lines and curves
	Gouraud + Edges	shademode + O	Combines gouraud shading and wireframe lines and curves
	Materials	materials	Opens the Drawing Explorer - Materials
	Light List	lightlist	Opens the Drawing Explorer - Lights
	Sun Properties	sunproperties	Opens the sun properties editor
	Geographic Location	geographiclocation	Opens the Geographic Location dialog

(*) Set the DISPSILH system variable On to display silhouette lines instead of a polygon mesh for curved surfaces.

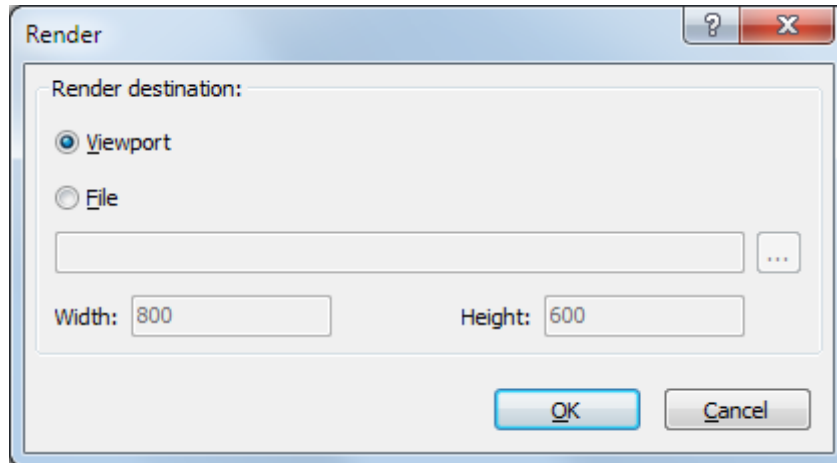


To render a viewport

1. Do one of the following

- Click the *Render* tool button () on the *Rendering* toolbar.
- Type *render* in the command bar, then press Enter.

The *Render* dialog displays:



2. Click the *OK* button.
The view is rendered.

The background of a rendered view is defined by the Background property of the view.

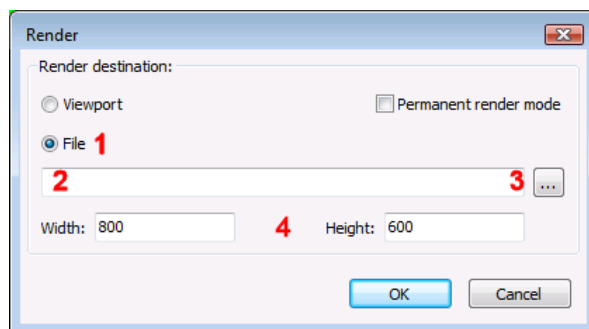
To save a rendering to a file

1. Do one of the following

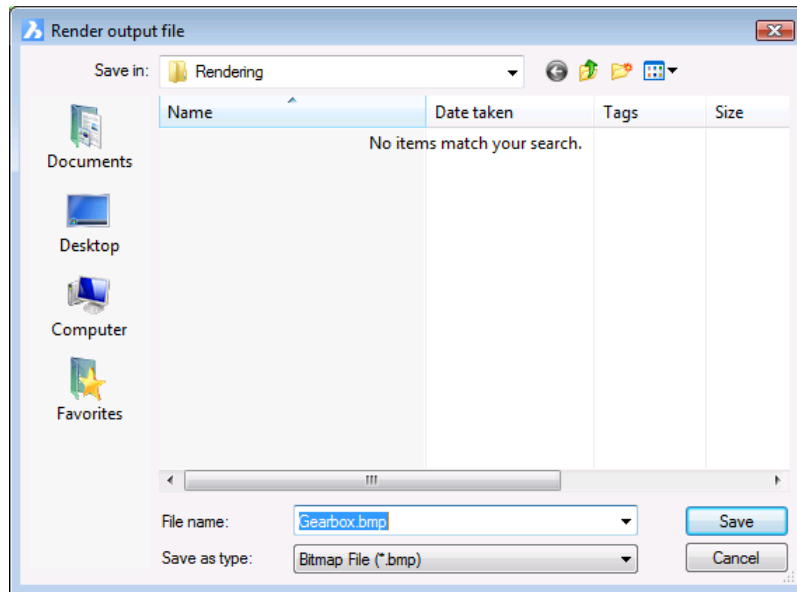
- Click the *Render* tool button () on the *Rendering* toolbar.
- Type *render* in the command bar, then press Enter.

The *Render* dialog displays.

2. Click the *File* radio button (1) on the *Render* dialog.



3. Type a name for the output file in the *name* field (2).
4. Click the *browse* button (3).
The *Render Output File* dialog displays.



5. Select an output folder in the *Save in* field.
6. Click the *Save as type* list button and select an image file type: *bmp*, *jpeg* or *png*.
7. Click the *Save* button.
The *Render Output File* dialog closes.
8. Type the desired resolution for the output file in the *Width* and *Height* fields (4).
9. Click the *OK* button to create the file.

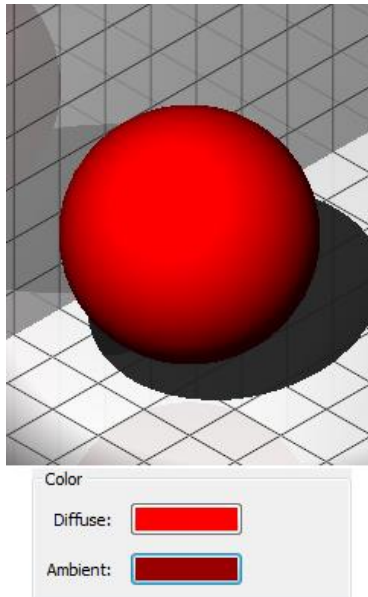
Rendering Materials

Command: MATERIALS

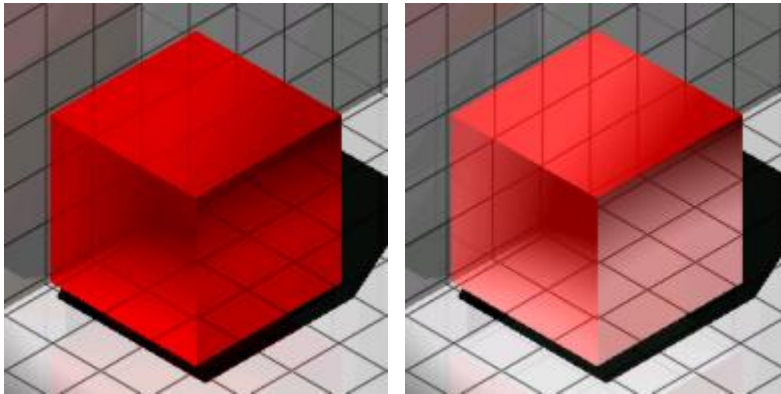
Material definitions are saved in the drawing. Each drawing contains the *Global* material. It is not possible to delete or rename the *Global* material. You can edit the properties of the *Global* material though.

Materials glossary

- **Diffuse color:** the color that the object reveals under pure direct, white light. It is perceived as the color of the object itself.
- **Ambient color:** the color of the object in the shadow, under ambient light rather than direct light.

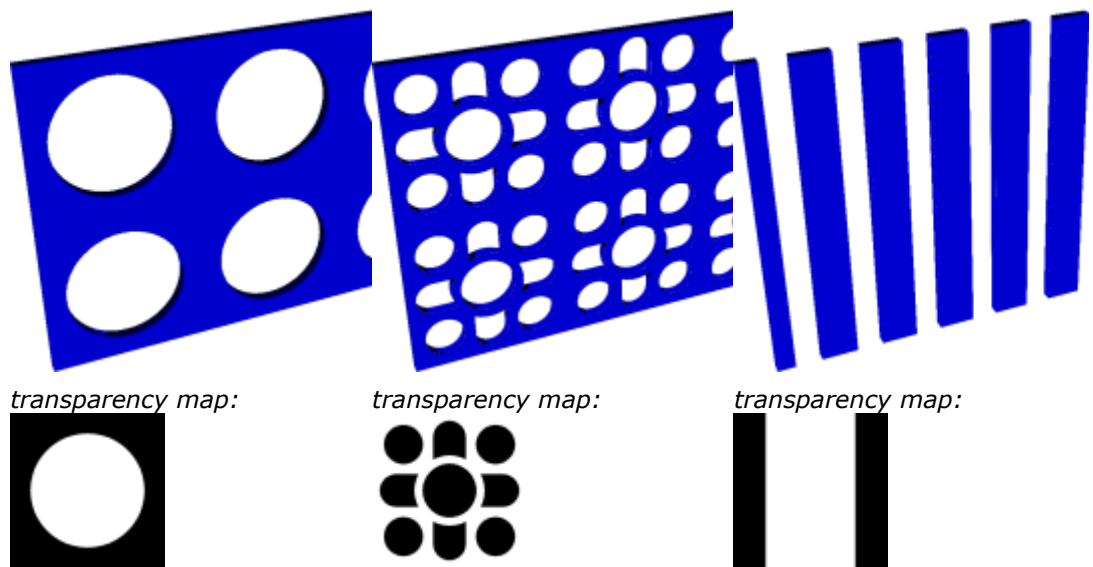


- **Highlights:** highly reflective highlights in the color of the material.



Metallic Highlights (left) and Non-metallic Highlights (right)

- **Shading:** how the color and brightness of a surface varies with lighting.
- **Texture-Mapping:** adds detail to a surface which is not included in the 3D model itself.
- **Diffuse map:** adds a structure or pattern to a surface to simulate a material.
- **Transparency map:** sets the transparency of a texture from 0% (black) to 100% (white) and anything between (grey scales).



- **Bump map:** simulates a certain roughness or bumpiness on surfaces.

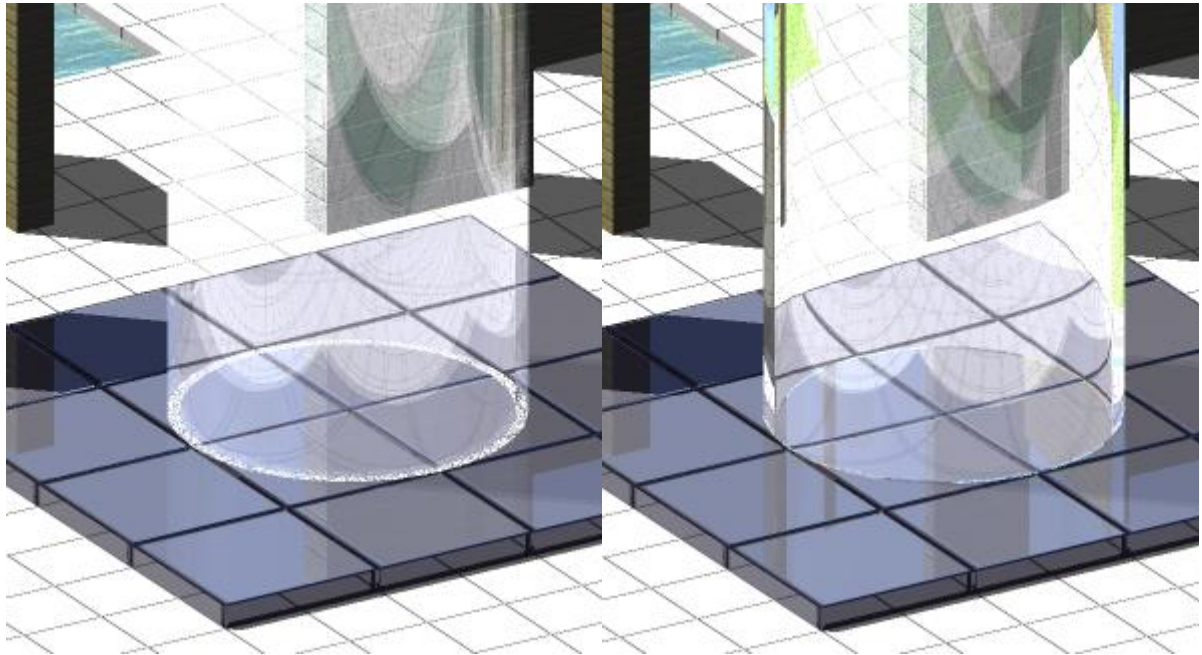


brick pattern: diffuse map only



brick pattern: diffuse map and bump map applied

- **Reflectivity:** the property of a surface to reflect light.
- **Transparency:** sharp transmission of light through solid objects.
- **Translucency:** scattered transmission of light through solid objects.
- **Refraction:** bending of light when passing through transparent objects.



Refraction index = 1.000

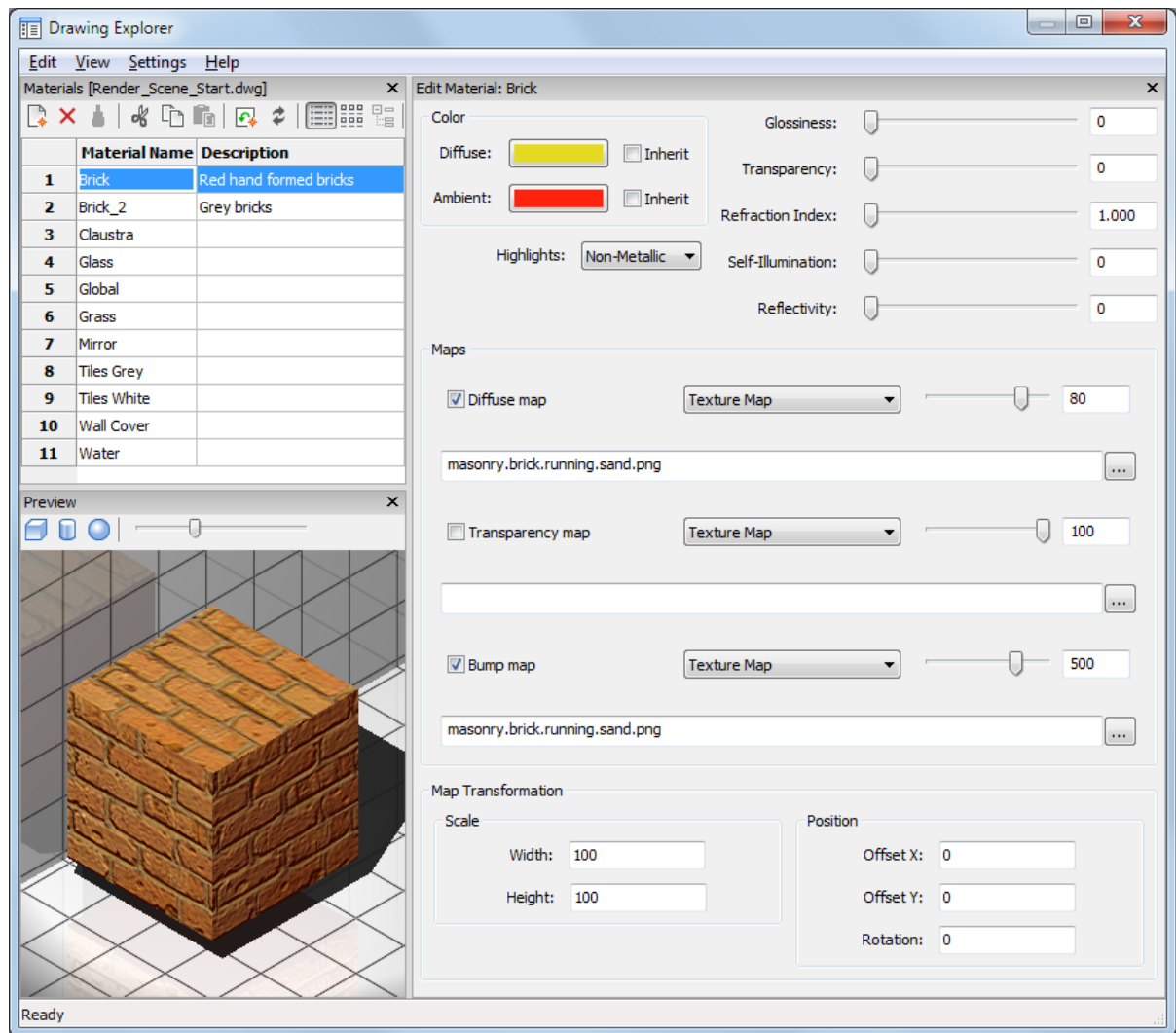
Refraction index = 2.000

- **self-illumination:** it appears as if a surface is emitting its own light. No light is cast on other objects.

Defining materials

1. Do one of the following:
 - Click the *Materials* tool button (🧱) on the *Rendering* toolbar.
 - Choose *Drawing Explorer > Materials* in the *Tools* menu.
 - Type *materials* in the command bar, then press Enter.

The *Drawing Explorer - Materials* window opens.
2. If necessary click the *Detail View* tool button (📐) or choose *Details* in the *View* menu on the *Drawing Explorer* window.
3. Click the *New* button (📄+) in the *Drawing Explorer* toolbar.
A new material is added.
4. Type a name for the new material in the *Name* field.
5. (option) Type a description in the *Description* field.
6. Do one of the following to define the color of the material:
 - Check the *Inherit* option to use the entity color.
 - Click the colored tile to open the *Select color* dialog.
7. Set the *Highlights* type: *Metallic* or *Non-Metallic*.
8. Define the properties of the material:



Assigning materials

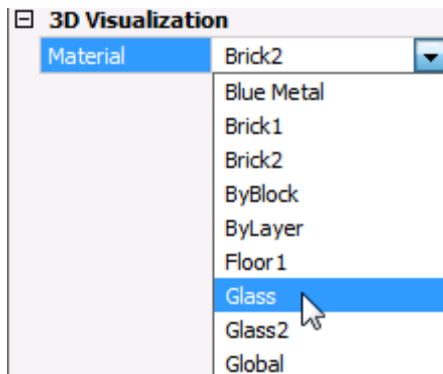
Materials can be assigned to a layer and to entities. If the Material property of an entity is BYLAYER. The material assigned to the layer of the entity will be used.

By default new entities get the BYLAYER material setting.

New layers get the Global material assigned.


To assign a material to an entity

1. Select the entity.
2. In the Properties bar, under *3D Visualization > Material*, click the down arrow and select the material in the *Material* list.




To assign a material to a layer

3. Do one of the following

- Click the *Explore Layers* tool button () in the *Entity Properties* toolbar.
- Choose *Layers > Layers...* in the *Settings* menu.
- Type *layer* in the command bar, then press Enter.

The *Drawing Explorer - Layers* dialog opens.

4. In the *Drawing Explorer - Layers* dialog click the *Detail view* tool button ()
5. Click the *Material* column of the layer you want to assign a material to.
6. Click the down arrow and select the material in the *Material* list.

Lights

Commands: DISTANTLIGHT, GEOGRAPHICLOCATION, LIGHT, LIGHTLIST, POINTLIGHT, SPOTLIGHT, SUNPROPERTIES, WEBLIGHT

Lighting is the key to make the viewer believe to look at a realistic scene.

Lighting glossary

- **Ambient light:** light from an unspecified source.
- **Back light:** light that adds depth and dimension by creating highlights.
- **Distant light:** light that is cast evenly throughout a scene with shadows all in the same direction.
- **Fall-off:** the attenuation or decrease in brightness the further from the light source.
- **Fill light:** the light that brightens dark areas and softens shadows from the main light.
- **Point light:** light that shines in all directions from a central spot.
- **Spot light:** focusable light that is aimed in one area.
- **Photometric web:** a 3D representation of the light intensity distribution of a light source.
- **Web light:** approximates real-world light distribution using a 3D representation of the light intensity. Web lights can be created only if the LIGHTINGUNITS system variable is set to 1 (American lighting units) or 2 (International lighting units).
- **Attenuation:** the fall-off in brightness the further from the light source.

General properties

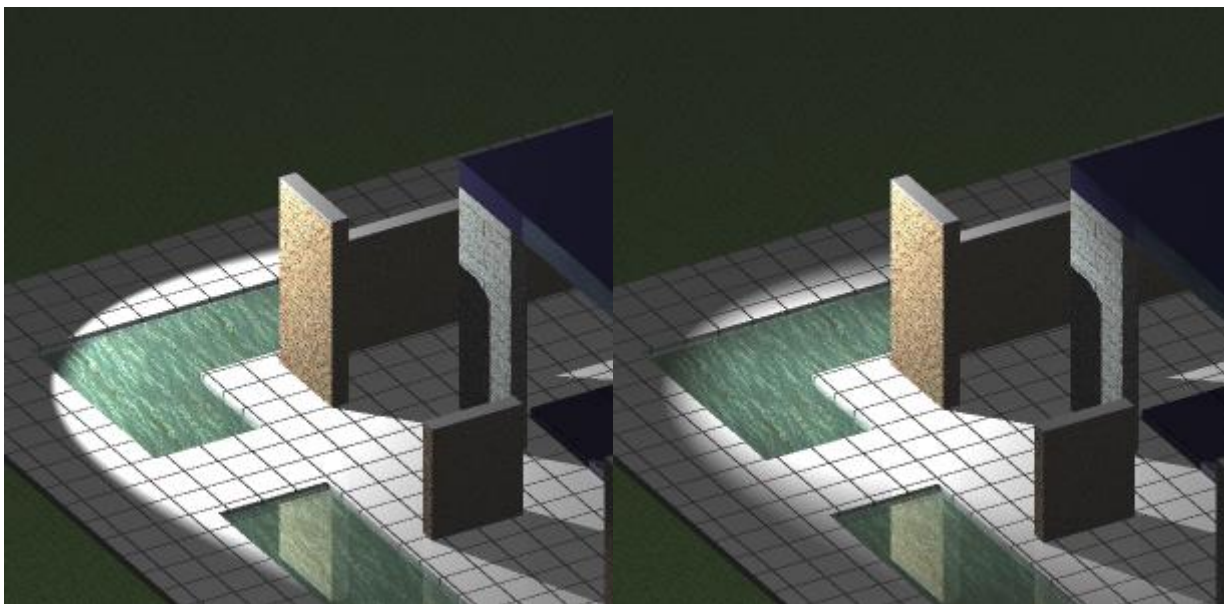
The general properties are common to all lights:

- **Name:** a user defined name for the light.
- **Type:** defines the type of light: spot, point, web or distant.
- **On/Off:** specifies whether the light is turned on or off.
- **Shadows:** specifies whether the light casts shadows or not. Turning shadows off to increases performance.
- **Intensity factor:** multiplies the *Lamp Intensity* property, the result is the *Resulting Intensity*.
- **Filter color:** defines the color of the light.
- **Plot glyph:** if on, the light glyphs are plotted.

Spotlight hotspot and falloff properties

When a surface is illuminated by a spotlight, there is an area of maximum illumination (hotspot) that is surrounded by an area of lesser intensity (falloff).

- **Hotspot angle:** angle of the central light cone (defines the hotspot)
- **Falloff angle:** angle of the full light cone



Hotspot angle = 55, Falloff angle = 60

Hotspot angle = 30, Falloff angle = 60

The difference between the hotspot angle and the falloff angle defines the area of lesser light intensity.

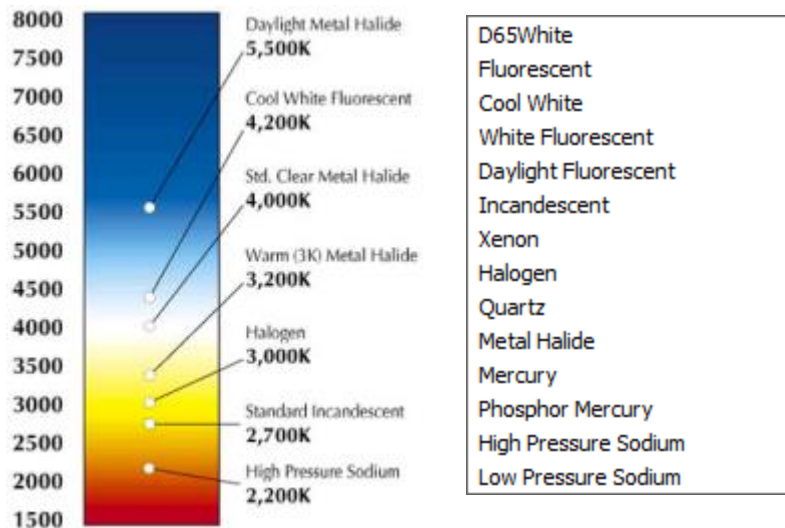
If the falloff angle and the hotspot angle are almost equal the edge of light cone is rather sharp (left).

The greater the difference between both angles, the softer the edge of the light cone (right).

Photometric properties

(not yet implemented)

- **Lamp intensity:** Defines the brightness of the light. The lamp intensity is expressed in candela (cd) which is the SI base unit of luminous intensity: the power emitted by a light source in a particular direction, weighted by the luminosity function (a standardized model of the sensitivity of the human eye to different wavelengths, also known as the luminous efficiency function)
- **Resulting intensity:** the product of the lamp intensity and intensity factor.
- **Lamp color:** defines the inherent color of the light or color temperature in Kelvin or standards.



color temperatures

lamp color list

- **Resulting color:** this is the final color of the light: a combination of the lamp color and the filter color.
- **Photometric web:** If the light type is *Web*, *Photometric Web* and *Web offsets* are available (not implemented yet).

Geometry properties

- **Position:** controls the location of the light.
- **Target:** defines the target point for point lights, spotlight and weblights
- **Targeted:** switches the *target* property on/off.

Attenuation properties

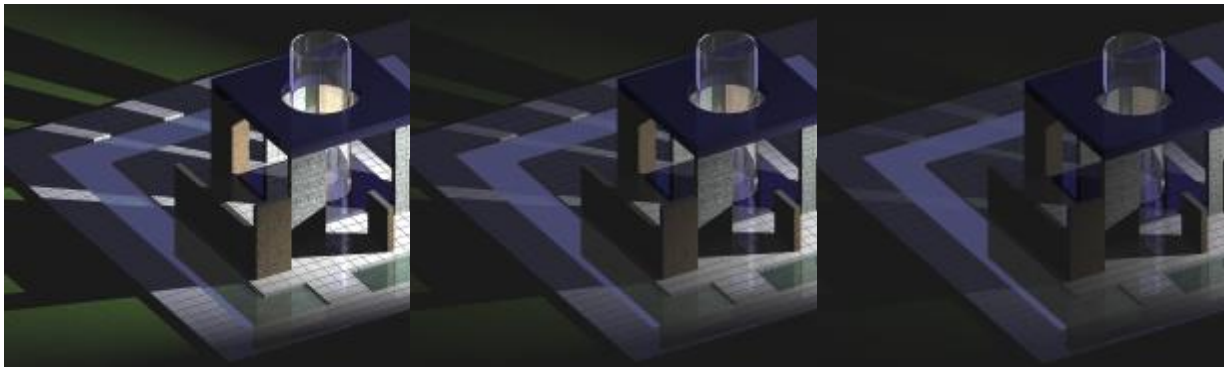
In the real world, an object appears darker if it is farther away from the light source. The attenuation properties define how the light diminishes over distance. Attenuation does not apply to photometric lights.

An alternative method to control the distance a light shines is the use of limits, which define the points from where a light starts to shine and where it stops. Using limits decreases the time needed to calculate the illumination of a scene.

Attenuation properties apply to spot lights and point lights.

- **Type:** the options are *none*, *inverse linear* and *inverse square*
- **None:** no attenuation. The distance to the light source has no influence.
- **Inverse Linear:** the attenuation is the inverse of the linear distance from the light: at a distance of 2 units from the light source, light is half as strong; at a distance of 4 units, light is one quarter as strong.

- **Inverse Square:** the attenuation is the inverse of the square of the distance from the light: at a distance of 2 units, light is one quarter as strong; at a distance of 4 units, light is one sixteenth as strong.
- **Use Limits:** if this property is set to Yes, the *Start limit offset* and *End limit offset* fields become active.
- **Start limit offset:** defines the point where the light starts to shine, measured from the center of the light.
- **End limit offset:** defines the point where the light stops to shine, measured from the center of the light.



No attenuation

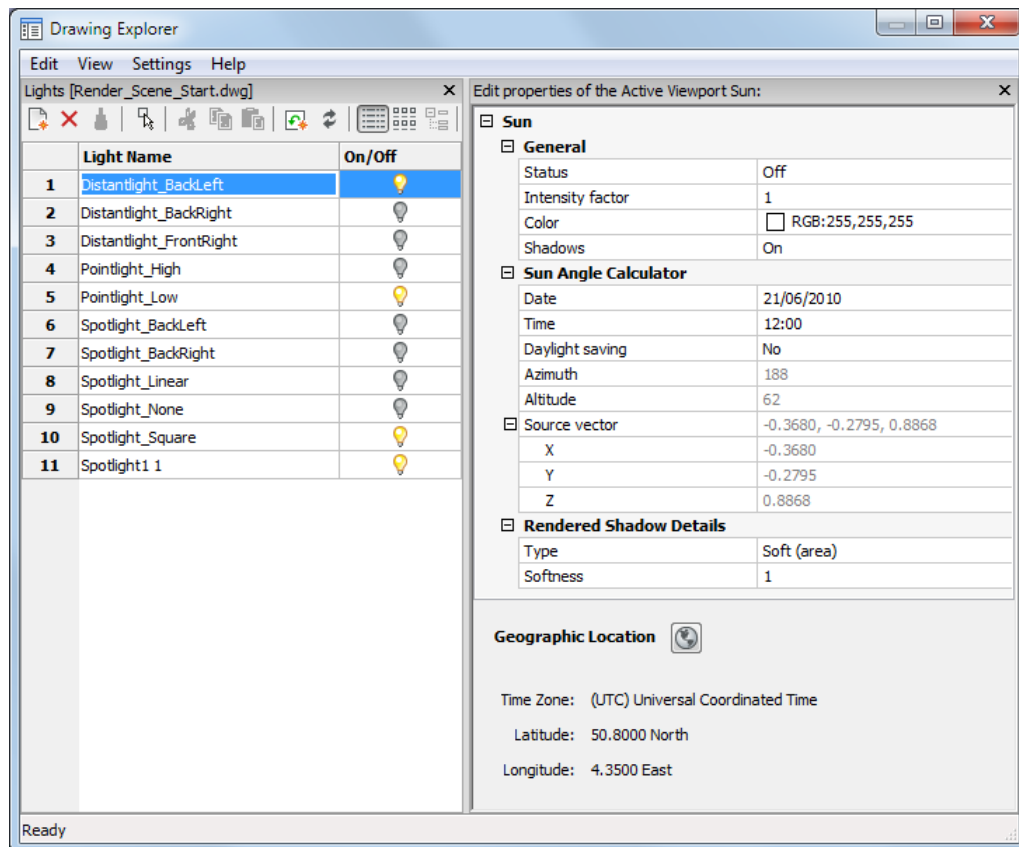
Inverse linear attenuation

Inverse square attenuation

To open the light list in the Drawing Explorer

Do one of the following:

- Click the *Light List...* button (🔍) on the *Lights* toolbar.
- Choose *Drawing Explorer > Lights* in the *Tools* menu.
- Choose *Light List..* in the *View | Rendering | Lights* menu
- Type *lightlist* or *LL* in the command bar, then press Enter.



Icon	Tool name	Description
	New light	Closes the <i>Drawing Explorer - Lights</i> dialog, then guides you to the creation of a new light in the drawing.
	Delete light	Deletes the selected light.
	Select in drawing	Closes the <i>Drawing Explorer - Lights</i> dialog and selects the light's glyph in the drawing. The properties of the light display in the Properties Bar.
	Light ON	Click to switch light off.
	Light OFF	Click to switch light on.

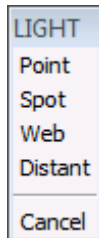
To define a light

1. Do one of the following:

- Type *light* in the command bar, then press Enter.




The command bar reads: Enter light type [Point/Spot/Web/Distant] <Point>:

A prompt menu displays:



Choose a light type in the prompt menu.

- Click the tool button of the light type you want to create on the *Lights* toolbar:

-  New spot light
-  New point light
-  New distant light

2. Do one of the following:

- Specify the source position in the drawing.
- Type the coordinates of the source position in the command bar and press Enter.
- Press Enter to accept the default coordinates.

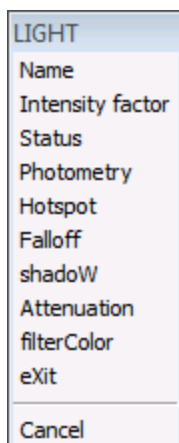
The command bar reads: Specify target position <0,0,-10>:

3. Do one of the following:

- Specify the target position in the drawing.
- Type the coordinates of the target position in the command bar and press Enter.
- Press Enter to accept the default coordinates.

The command bar reads: Enter an option to change: Name/Intensity factor/Status/Photometry/Hotspot/Falloff/shadow/Attenuation/filterColor/<eXit>:

A prompt menu displays:



4. Do one of the following:

- Select *Name* in the prompt box.
- Type *N* in the command bar and press Enter.

The command bar reads: Enter light name <Spotlight1>:

5. Do one of the following:
 - Type a name in the command bar and press Enter.
 - Press Enter to accept the default name.

The command bar reads: Enter an option to change: Name/Intensity factor/Status/Photometry/Hotspot/Falloff/shadow/Attenuation/filterColor/<eXit>:

6. Do one of the following:
 - Click a light property in the prompt menu.
 - Type the capitalized letter of a light property in the command bar and press Enter.

You are prompted to define the selected property in the command bar.

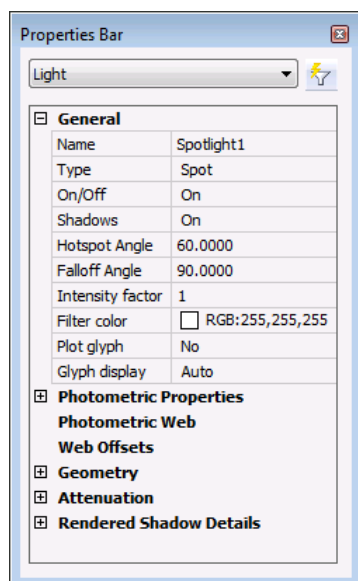
7. Do one of the following to conclude the creation of the light:
 - Choose Exit in the prompt menu.
 - Press Enter.

NOTE It is not possible to define a web light if the *LIGHTINGUNITS* system variable is zero (No lighting units).

To edit a light


1. Select the light in the drawing.
If the Properties bar is not open yet, double click the light.

The properties of the selected light display in the Properties Bar:

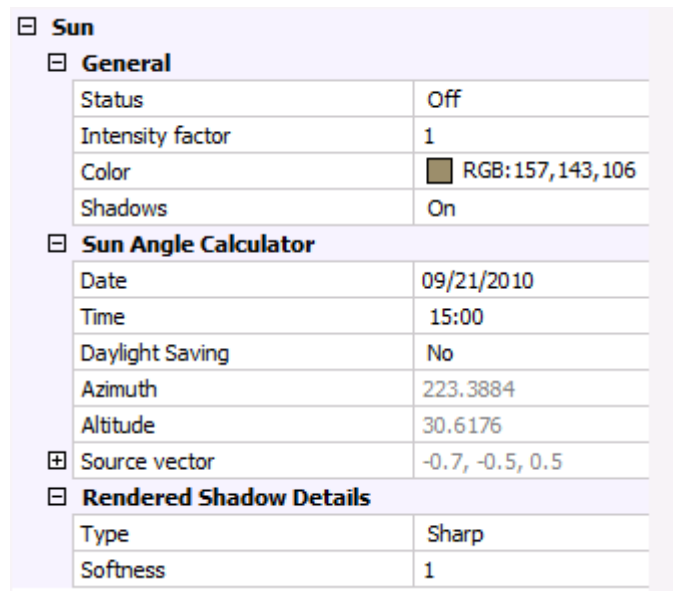



2. To edit a property:
 - Expand the settings group if needed.
 - Select the property
 - Edit the selected property.

To define the sun properties

- Do one of the following:
 - Click the *Sun Properties...* tool button () on the *Render* toolbar.
 - Type *sunproperties* or *sun* in the command bar, then press Enter.


The *Drawing Explorer - Lights* dialog displays, showing the sun properties in the *Editor* pane.



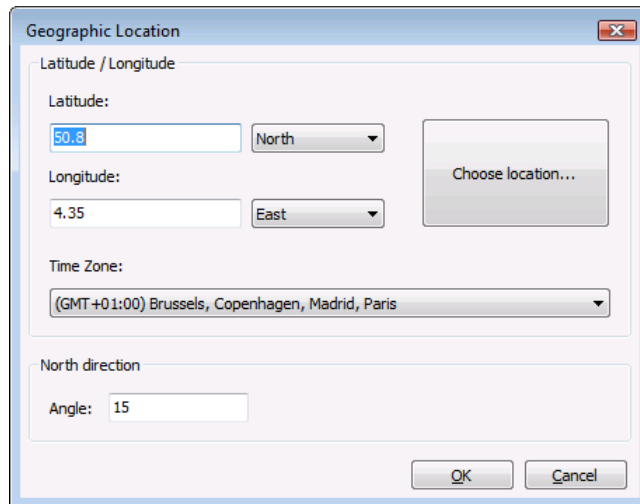
Sun	
General	
Status	Off
Intensity factor	1
Color	 RGB:157,143,106
Shadows	On
Sun Angle Calculator	
Date	09/21/2010
Time	15:00
Daylight Saving	No
Azimuth	223.3884
Altitude	30.6176
Source vector	-0.7, -0.5, 0.5
Rendered Shadow Details	
Type	Sharp
Softness	1

- To edit a property:
 - Expand the settings group if needed.
 - Select the property
 - Edit the selected property.

To define the geographic location

- Do one of the following:
 - Click the *Geographic location...* tool button () on the *Render* toolbar.
 - Click the *Geographic location...* tool button () on *Editor* pane of the *Drawing Explorer - Lights* dialog.
 - Type *geographiclocation* or *geo* in the command bar, then press Enter.

The *Geographic Location* dialog displays:



Geographic Location

Latitude / Longitude

Latitude: North

Longitude: East

Time Zone: (GMT+01:00) Brussels, Copenhagen, Madrid, Paris

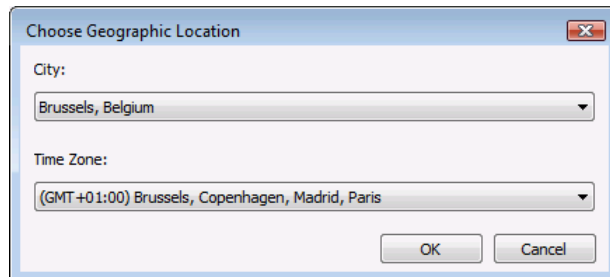
North direction

Angle:

Choose location...

OK Cancel

2. Do one of the following:
 - Type the latitude and longitude of the location in the *Latitude* and *Longitude* fields and select the appropriate hemisphere for each setting.
 - Click the *Choose location...* button, then select a city and timezone and click *the* OK button.



Choose Geographic Location

City: Brussels, Belgium

Time Zone: (GMT+01:00) Brussels, Copenhagen, Madrid, Paris

OK Cancel

3. Choose a time zone in the *Time Zone* list.
4. Define the direction of the north.
5. Click the *OK* button.